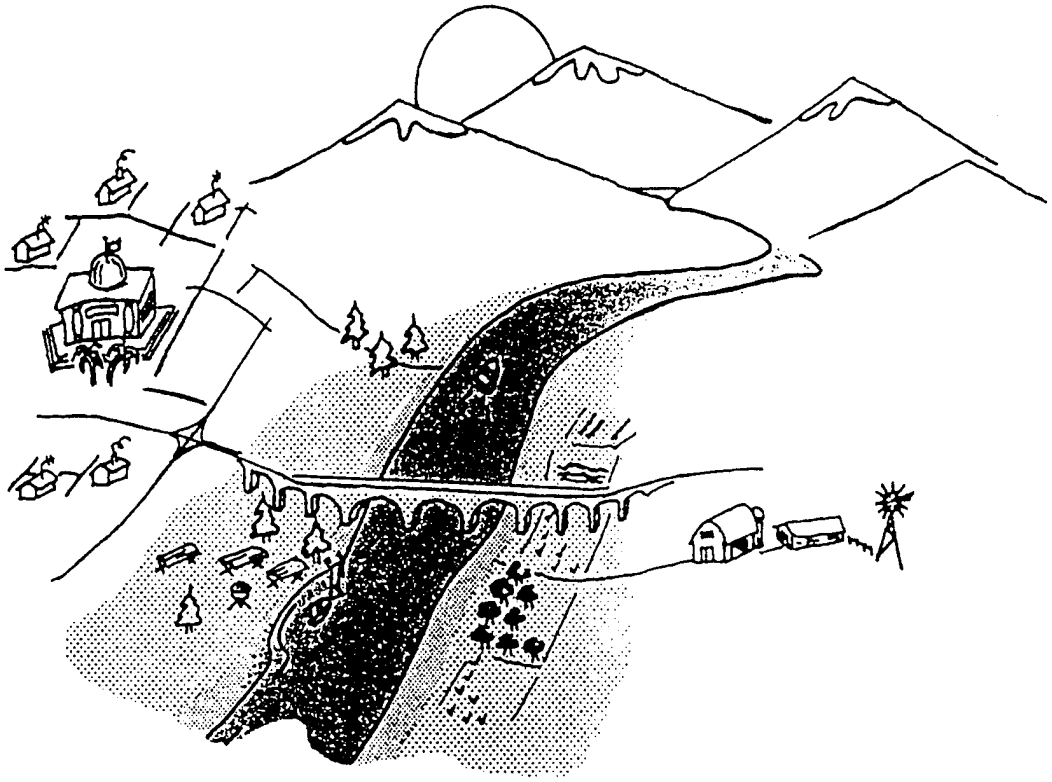
US Army Corps
of Engineers
Hydrologic Engineering Center

HEC-RAS
River Analysis System



User's Manual

19960807 019

Version 1.0
July 1995

DTIC QUALITY INSPECTED 1

CPD-68

Approved for Public Release. Distribution is unlimited.

Cover photo adapted from:

Flood Plain Management Program, Handbook for Public Officials
Department of Water Resources
State of California
August 1986

HEC-RAS

River Analysis System

User's Manual

Version 1.0

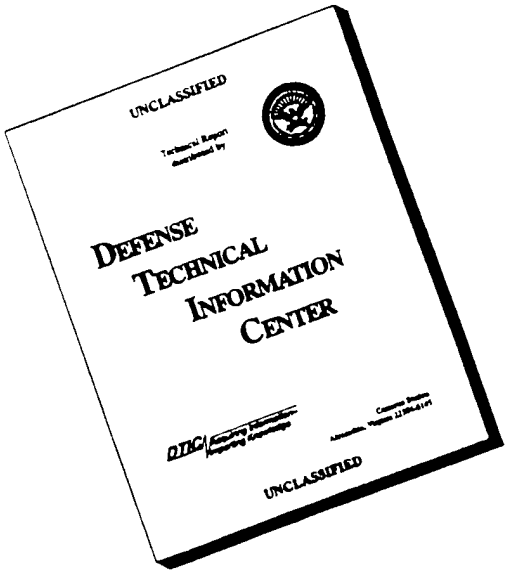
July 1995

US Army Corps of Engineers
Hydrologic Engineering Center
609 Second Street
Davis, CA 95616

(916) 756-1104
(916) 756-8250 FAX

CPD-68

DISCLAIMER NOTICE



THIS DOCUMENT IS BEST QUALITY AVAILABLE. THE COPY FURNISHED TO DTIC CONTAINED A SIGNIFICANT NUMBER OF PAGES WHICH DO NOT REPRODUCE LEGIBLY.

Table of Contents

Foreword iv

Chapter 1 Introduction 1-1
General Philosophy of the Modeling System 1-2
Overview of Program Capabilities 1-2
User Interface 1-2
Hydraulic Analysis Components 1-3
Data Storage and Management 1-4
Graphics and Reporting 1-4
HEC-RAS Documentation 1-5
Overview of This Manual 1-5

Chapter 2 Installing HEC-RAS 2-1
Hardware and Software Requirements 2-2
Installation Procedure 2-2

Chapter 3 Working With HEC-RAS - An Overview 3-1
Starting HEC-RAS 3-2
Steps in Developing a Hydraulic Model With HEC-RAS 3-3
Starting a New Project 3-4
Entering Geometric Data 3-4
Entering Flow and/or Sediment Data 3-7
Performing The Hydraulic Computations 3-8
Viewing and Printing Results 3-10
Importing HEC-2 Data 3-14
Getting and Using Help 3-17

Chapter 4 Example Application 4-1
Starting a New Project 4-2
Entering Geometric Data 4-3
Drawing the Schematic of the River System 4-3
Entering Cross Section Data 4-4
Entering Junction Data 4-9
Saving the Geometry Data 4-10
Entering Steady Flow Data 4-10
Performing the Hydraulic Calculations 4-13
Viewing Results 4-14
Printing Graphics and Tables 4-20
Sending Graphics Directly to the Printer 4-20
Sending Graphics to the Windows Clipboard 4-20
Sending Tables Directly to the Printer 4-21
Sending Tables to the Windows Clipboard 4-21
Exiting the Program 4-22

Chapter 5 Working With Projects 5-1
Understanding Projects 5-1
Elements of a Project 5-2
Plan Files 5-2
Run Files 5-2
Output Files 5-3
Geometry Files 5-3
Steady Flow Data Files 5-3
Unsteady Flow Data Files 5-3
Sediment Data Files 5-4
Hydraulic Design Data Files 5-4
Creating, Opening, Saving, Renaming, and Deleting Projects 5-5
Project Options 5-5

Chapter 6 Entering and Editing Geometric Data 6-1
Developing the River System Schematic 6-1
Building the Schematic 6-1
Editing the Schematic 6-2
Interacting With the Schematic 6-3
Cross Section Data 6-4
Entering Cross Section Data 6-4
Editing Cross Section Data 6-6
Cross Section Options 6-7
Plotting Cross Section Data 6-14
Stream Junctions 6-14
Entering Junction Data 6-14
Selecting a Modeling Approach 6-15
Bridges and Culverts 6-16
Cross Section Locations 6-16
Contraction and Expansion Losses 6-18
Bridge Hydraulic Computations 6-18
Entering and Editing Bridge Data 6-21
Culvert Hydraulic Computations 6-30
Entering and Editing Culvert Data 6-31
Bridge and Culvert Options 6-35
Bridge and Culvert View Features 6-36
Multiple Bridge and/or Culvert Openings 6-37
Entering Multiple Opening Data 6-38
Defining The Openings 6-41
Multiple Opening Calculations 6-42
Cross Section Interpolation 6-44
Viewing and Editing Data Through Tables 6-50
Manning's n values 6-50
Reach Lengths 6-51
Contraction and Expansion Coefficients 6-51
Saving the Geometric Data 6-53

Chapter 7 Performing a Steady Flow Analysis 7-1
Entering and Editing Steady Flow Data 7-1
Steady Flow Data 7-1
Boundary Conditions 7-2
Steady Flow Data Options 7-4
Saving the Steady Flow Data 7-5
Performing Steady Flow Calculations 7-6
Defining a Plan 7-6
Saving the Plan Information 7-7
Simulation Options 7-7
Starting the Computations 7-11

Chapter 8 Viewing Results 8-1
Cross Sections, Profiles, and Rating Curves 8-1
Viewing Graphics on the Screen 8-1
Graphical Plot Options 8-4
Plotting Velocity Distribution Output 8-5
Plotting Other Variables in Profile 8-6
Plotting One Variable Versus Another 8-7
Sending Graphics to the Printer or Plotter 8-8
Sending Graphics to the Windows Clipboard 8-9
X-Y-Z Perspective Plots 8-10
Tabular Output 8-11
Cross Section Tables 8-11
Cross Section Table Options 8-14
Profile Tables 8-15
User Defined Output Tables 8-17
Sending Tables to the Printer 8-18
Sending Tables to the Windows Clipboard 8-19
Viewing Results From the River System Schematic 8-20

Chapter 9 Performing a Floodway Encroachment Analysis 9-1
General 9-2
Entering Floodway Encroachment Data 9-2
Performing the Floodway Encroachment Analysis 9-5
Viewing the Floodway Encroachment Results 9-6

Chapter 10 Trouble Shooting With HEC-RAS 10-1
Built in Data Checking 10-1
Checking The Data as it is Entered 10-1
Checking Data Before Computations are Performed 10-2
Errors, Warnings, and Notes 10-3
Log Output 10-5
Setting Log File Output 10-5
Viewing The Log File 10-6

Reviewing and Debugging the Normal Output 10-7
Viewing Graphics 10-7
Viewing Tabular Output 10-7
The Occurance of Critical Depth 10-7

Appendix A References A-1

Foreword

The HEC-RAS software was developed at the Hydrologic Engineering Center (HEC). The software was designed by Mr. Gary W. Brunner, leader of the HEC-RAS development team. The user interface and graphics were programmed by Mr. Mark R. Jensen. The steady flow water surface profiles module was programmed by Mr. Steven S. Piper. The routines that import HEC-2 data were developed by Ms. Joan Klipsch. The cross section interpolation routines were developed by Mr. Alfredo Mantalvo.

Many of the HEC staff made contributions in the development of this software, including: Vern R. Bonner, Richard Hayes, John Peters, and Michael Gee. Mr. Darryl Davis was the director during the development of this software..

This manual was written by Mr. Gary W. Brunner.

CHAPTER 1

Introduction

Welcome to the Hydrologic Engineering Center's River Analysis System (HEC-RAS). This software allows you to perform one-dimensional steady flow, unsteady flow, and sediment transport calculations (The current version of HEC-RAS can only perform steady flow calculations. Unsteady flow and sediment transport will be added in future versions).

The HEC-RAS modeling system was developed as a part of the Hydrologic Engineering Center's "Next Generation" (NexGen) of hydrologic engineering software. The NexGen project encompasses several aspects of hydrologic engineering, including: rainfall-runoff analysis; river hydraulics; reservoir system simulation; flood damage analysis; and real-time river forecasting for reservoir operations.

This chapter discusses the general philosophy of HEC-RAS and gives you a brief overview of the capabilities of the modeling system. Documentation for HEC-RAS is discussed, as well as an overview of this manual.

Contents

- General Philosophy of the Modeling System
- Overview of Program Capabilities
- HEC-RAS Documentation
- Overview of This Manual

General Philosophy of the Modeling System

HEC-RAS is an integrated system of software, designed for interactive use in a multi-tasking, multi-user network environment. The system is comprised of a graphical user interface (GUI), separate hydraulic analysis components, data storage and management capabilities, graphics and reporting facilities.

The system contains three one-dimensional hydraulic analysis components for: (1) steady flow water surface profile computations; (2) unsteady flow simulation; and (3) movable boundary sediment transport computations. A key element is that all three components use a common geometric data representation and common geometric and hydraulic computation routines. In addition to the three hydraulic analysis components, the system contains several hydraulic design features that can be invoked once the basic water surface profiles are computed.

The current version of HEC-RAS only supports Steady Flow water surface profile calculations. New features and additional capabilities will be added in future releases.

Overview of Program Capabilities

HEC-RAS is designed to perform one-dimensional hydraulic calculations for a full network of natural and constructed channels. The following is a description of the major capabilities of HEC-RAS.

User Interface

The user interacts with HEC-RAS through a graphical user interface (GUI). The main focus in the design of the interface was to make it easy to use the software, while still maintaining a high level of efficiency for the user. The interface provides for the following functions:

- File management
- Data entry and editing
- Hydraulic analyses
- Tabulation and graphical displays of input and output data
- Reporting facilities
- On-line help

Hydraulic Analysis Components

Steady Flow Water Surface Profiles. This component of the modeling system is intended for calculating water surface profiles for steady gradually varied flow. The system can handle a full network of channels, a dendritic system, or a single river reach. The steady flow component is capable of modeling subcritical, supercritical, and mixed flow regime water surface profiles.

The basic computational procedure is based on the solution of the one-dimensional energy equation. Energy losses are evaluated by friction (Manning's equation) and contraction/expansion (coefficient multiplied by the change in velocity head). The momentum equation is utilized in situations where the water surface profile is rapidly varied. These situations include mixed flow regime calculations (i.e. hydraulic jumps), hydraulics of bridges, and evaluating profiles at river confluences (stream junctions).

The effects of various obstructions such as bridges, culverts, weirs, and structures in the flood plain may be considered in the computations. The steady flow system is designed for application in flood plain management and flood insurance studies to evaluate floodway encroachments. Also, capabilities are available for assessing the change in water surface profiles due to channel improvements, levees, and ice cover.

Special features of the steady flow component include: multiple plan analyses; multiple profile computations; and multiple bridge and/or culvert opening analysis.

Unsteady Flow Simulation. This component of the HEC-RAS modeling system is capable of simulating one-dimensional unsteady flow through a full network of open channels. The unsteady flow equation solver has been adapted from Dr. Robert L. Barkau's UNET model (Barkau, 1992 and HEC, 1993). This unsteady flow component was developed primarily for subcritical flow regime calculations.

The hydraulic calculations for cross-sections, bridges, culverts, and other hydraulic structures that were developed for the steady flow component have been incorporated into the unsteady flow module. Additionally, the unsteady flow component has the ability to model storage areas, navigation dams, gated spillways, tunnels, pumping stations, and levee failures.

Sediment Transport/Movable Boundary Computations. This component of the modeling system is intended for the simulation of one-dimensional sediment transport/movable boundary calculations resulting from scour and deposition over moderate time periods (typically years, although applications to single flood events are possible).

The sediment transport potential is computed by grain size fraction, thereby allowing the simulation of hydraulic sorting and armoring. Major features include the ability to model a full network of streams, channel dredging, various levee and encroachment alternatives, and the use of several different equations for the computation of sediment transport.

The model is designed to simulate long-term trends of scour and deposition in a stream channel that might result from modifying the frequency and duration of the water discharge and stage, or modifying the channel geometry. This system can be used to evaluate deposition in reservoirs, design channel contractions required to maintain navigation depths, predict the influence of dredging on the rate of deposition, estimate maximum possible scour during large flood events, and evaluate sedimentation in fixed channels.

Data Storage and Management

Data storage is accomplished through the use of "flat" files (ASCII and binary). User input data are stored in flat files under separate categories of project, plan, geometry, steady flow, unsteady flow, and sediment data. Output data is predominantly stored in separate binary files.

Data management is accomplished through the user interface. The modeler is requested to enter a single filename for the project being developed. Once the project filename is entered, all other files are automatically created and named by the interface as needed. The interface provides for renaming, moving, and deletion of files on a project by project basis.

Graphics and Reporting

Graphics include X-Y plots of the river system schematic, cross-sections, profiles, rating curves, hydrographs, and many other hydraulic variables. A pseudo three-dimensional plot of multiple cross-sections is also provided. Tabular output is available. Users can select from pre-defined tables or develop their own customized tables. All graphical and tabular output can be displayed on the screen, sent directly to a printer (or plotter), or passed through the Windows clipboard to other software, such as a word-processor or spreadsheet.

Reporting facilities allow for printed output of input data as well as output data. Reports can be customized as to the amount and type of information desired.

HEC-RAS Documentation

The HEC-RAS package includes several documents. Each document is designed to help the modeler learn to use a particular aspect of the modeling system. The documentation has been broken up into the following three categories:

Documentation	Description
User's Manual	This manual is a guide to using HEC-RAS. The manual provides an introduction and overview of the modeling system, installation instructions, how to get started, simple examples, detailed descriptions of each of the major modeling components, and how to view graphical and tabular output.
Hydraulic Reference Manual	This manual describes the theory and data requirements for the hydraulic calculations performed by HEC-RAS. Equations are presented along with the assumptions used in their derivation. Discussions are provided on how to estimate model parameters, as well as guidelines on various modeling approaches.
Applications Guide	This document contains a series of examples that demonstrate various aspects of HEC-RAS. Each example consists of a problem statement, data requirements, general outline of solution steps, displays of key input and output screens, and discussions of important modeling aspects.

Overview of This Manual

This user's manual is the primary piece of documentation on how to use the HEC-RAS system. The manual is organized as follows:

- Chapters 1-2 provide an introduction and overview of HEC-RAS, as well as instructions on how to install the software.

- Chapters 3-5 describe how to use the HEC-RAS software in a step-by-step procedure, including a sample problem that the user can follow along with. Understanding how this system works with projects is also discussed.

- Chapters 6-7 explain in detail how to enter and edit data, and how to perform the different types of analyses that are available.

- Chapter 8 provides detailed discussions on how to view graphical and tabular output, as well as how to develop user defined tables.

- Chapter 9 describes how to perform a floodway encroachment analysis.

- Appendix A contains a list of references.

- Appendix B contains a listing of all the messages (errors, warnings, and notes) that come from the HEC-RAS package (not finished yet).

- Appendix C provides a discussion on "Trouble Shooting" (not finished yet).

CHAPTER 2

Installing HEC-RAS

You install HEC-RAS using the program SETUP.EXE. The Setup program installs the HEC-RAS programs, sample applications, and the Help system.

This chapter discusses the hardware and system requirements needed to use HEC-RAS, and how to install the software.

Contents

- Hardware and Software Requirements
- Installation Procedure

Important

You cannot simply copy files from the distribution disks to your hard disk and run HEC-RAS. You must use the Setup program, which decompresses and installs the files to the appropriate directories.

Hardware and Software Requirements

Before you install the HEC-RAS software, make sure that your computer has the minimum required hardware and software. In order to get the maximum performance from the HEC-RAS software, recommended hardware and software is show in parenthases. This version of HEC-RAS will run on a microcomputer that has the following:

- Any IBM or compatible machine with an 80386 processor or higher (a 80486 or higher is recommended).
- A hard disk with at least 10 megabytes of free space (20 megabytes or more is recommended).
- A 3 1/2" floppy drive.
- A minimum of 4 megabytes of RAM (8 or more is recommended).
- A mouse.
- Color VGA or better Video Display (Recommend running in Super VGA or higher, and as large a monitor as possible)
- MS-DOS version 3.3 or later, and MS Windows version 3.1 or later **(running in 386 enhanced mode)**. This software will also run under MicroSoft Windows NT version 3.5 or later.

Installation Procedure

Installation of the HEC-RAS software is accomplished through the use of the Setup program. When you run the Setup program, you will be asked to set a path for the program and data files. A suggested directory of "\HECRAS" will be provided. You may choose to use this directory name or provide one of your own.

To install the software onto your hard disk, do the following:

1. Start Windows by typing **WIN**, then press the **ENTER** key.
2. Insert Disk 1 into the A drive (or B if necessary).
3. From the **File** menu of the Windows Program Manager, choose **Run**.

4. Type **a:setup** (or **b:setup** if disk 1 is in the B drive) and press **ENTER**.

5. Follow the setup instructions on the screen.

Important

Once you have finished installing the HEC-RAS software, you will need to re-start Windows in order for the program to function properly.

CHAPTER 3

Working With HEC-RAS - An Overview

HEC-RAS is an integrated package of hydraulic analysis programs, in which the user interacts with the system through the use of a Graphical User Interface (GUI). The system is capable of performing Steady Flow water surface profile calculations, and will include Unsteady Flow, Sediment Transport, and several hydraulic design computations in the future.

In HEC-RAS terminology, a **Project** is a set of data files associated with a particular river system. The modeler can perform any or all of the various types of analyses, included in the HEC-RAS package, as part of the project. The data files for a project are categorized as follows: plan data, geometric data, steady flow data, unsteady flow data, and sediment data.

During the course of a study the modeler may want to formulate several different **Plans**. Each plan represents a specific set of geometric data and flow data. Once the basic data are entered into the HEC-RAS, the modeler can easily formulate new plans. After simulations are made for the various plans, the results can be compared in both tabular and graphical form.

This chapter provides an overview of how a study is performed with the HEC-RAS software. Special topics on how to import HEC-2 data and how to use on-line help are also covered.

Contents

- Starting HEC-RAS
- Steps in Developing a Hydraulic Model With HEC-RAS
- Importing HEC-2 Data
- Getting and Using Help

Starting HEC-RAS

When you run the HEC-RAS Setup program, you automatically get a new program group and program icon for HEC-RAS in Windows. They should appear as shown in Figure 3.1.

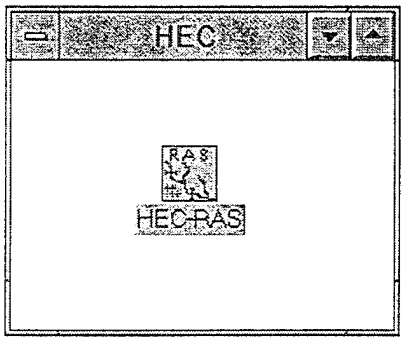


Figure 3.1 The HEC-RAS Icon in Windows

To Start HEC-RAS from Windows:

- Double-click on the HEC-RAS Icon.

When you first start HEC-RAS, you will see the main window as shown in Figure 3.2.

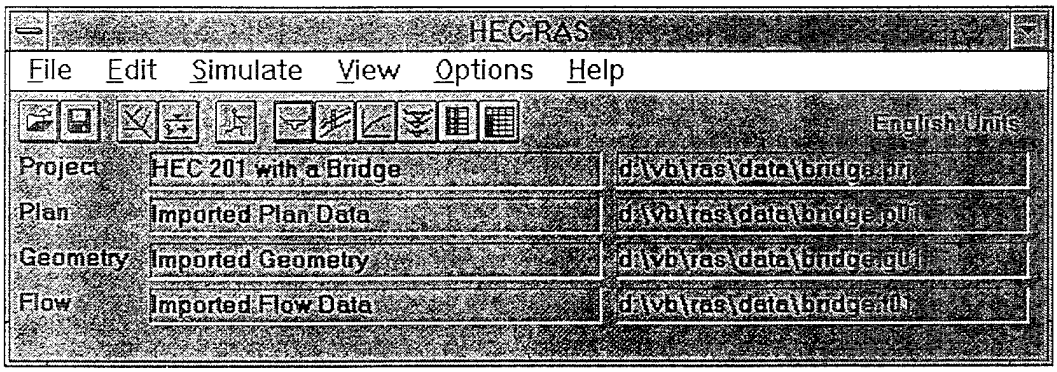


Figure 3.2 The HEC-RAS Main Window

The HEC-RAS main window has the following options on the menu bar.

File This option is used for file management. Options available under the File menu include: New Project; Open Project; Save Project; Save Project As; Rename Project: Delete Project; Import HEC-2 Data; Import HEC-RAS data; and Exit. In addition, the four most recently opened projects will be listed at the bottom of the File menu.

Edit This option is used for entering and editing data. Data are categorized into four types: Geometric Data; Steady Flow Data; Unsteady Flow Data; and Sediment Data.

Simulate This option is used to perform the hydraulic calculations. The options under this menu item include: Steady Flow Analysis; Unsteady Flow Analysis; Sediment Analysis; and Hydraulic Design Functions.

View This option contains a set of tools that provide for graphical and tabular displays of the model output. The View menu item currently includes: Cross Sections; Water Surface Profiles; Rating Curves; X-Y-Z Perspective Plots; Cross Section Tables; Profile Tables; and Summary Err, Warn, Notes.

Options This menu item allows the user to change Program Setup options; set Default Parameters; establish the Default Units System (English or Metric); and Convert Project Units (English to Metric, or Metric to English).

Help This option allows the user to get on-line help, as well as display the current version information about HEC-RAS.

Steps in Developing a Hydraulic Model with HEC-RAS

There are five main steps in creating a hydraulic model with HEC-RAS:

- Starting a new project
- Entering geometric data
- Entering flow and/or sediment data
- Performing the hydraulic calculations
- Viewing and printing results

Starting a New Project

The first step in developing a hydraulic model with HEC-RAS is to establish which directory you wish to work in and to enter a title for the new project. To start a new project, go to the **File** menu on the main HEC-RAS window and select **New Project**. This will bring up a New Project window as shown in Figure 3.3.

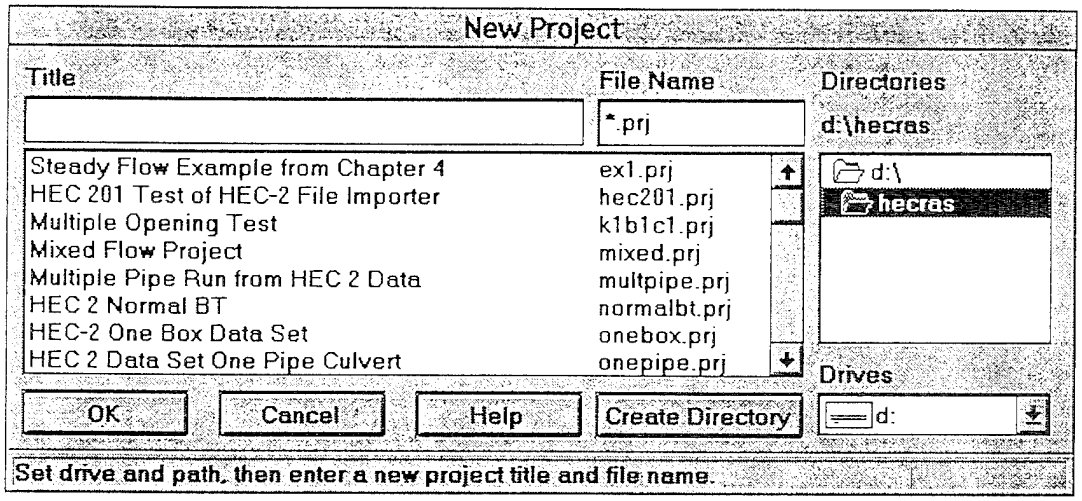


Figure 3.3 New Project window

As shown in Figure 3.3, you first select the drive and path that they want to work in (to actually select a path you must double click the directory you want in the directory box), then enter a project title and file name. The project filename must have the extension ".PRJ", the user is not allowed to change this. Once you have entered all the information, press the "**OK**" button to have the information accepted. After the OK button is pressed, a message box will appear with the title of the project and the directory that the project is going to be placed in. If this information is correct, press the **OK** button. If the information is not correct, press the **Cancel** button and you will be placed back into the **New Project** window.

Note: Before any Geometric data and Flow data are entered, the user should select the Units System (English or Metric) that they would like to work in. This is accomplished by selecting **Unit System** from the **Options** menu on the main HEC-RAS window.

Entering Geometric Data

The next step is to enter the necessary geometric data, which consist of connectivity information for the stream system (Stream Network), cross-section data, and hydraulic structure data (bridges, culverts, weirs, etc.). Geometric data are entered by selecting **Geometric Data** from the **Edit** menu on the main HEC-RAS window. Once this option is selected, the geometric

data window will appear as show in Figure 3.4 (except yours will be blank when you first bring this screen up for a new project).

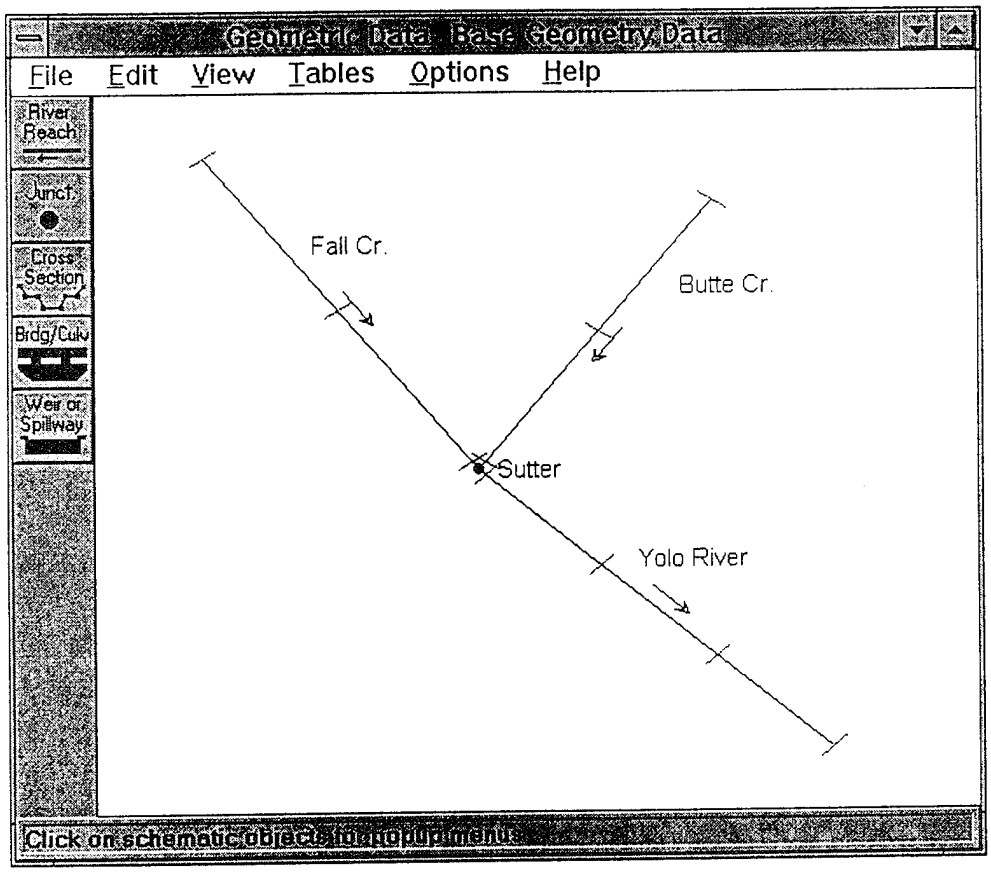


Figure 3.4 Geometric Data Window

The modeler develops the geometric data by first drawing in the river system schematic. This is accomplished, on a reach-by-reach basis, by pressing the **River Reach** button and then drawing in a reach from upstream to downstream (in the positive flow direction). After the reach is drawn, the user is prompted to enter an identifier for the reach. The identifier can be up to twelve characters in length. As reaches are connected together, junctions are automatically formed by the interface. The modeler is also prompted to enter an identifier for each junction.

After the river system schematic is drawn, the modeler can start entering cross-section and hydraulic structure data. Pressing the **Cross Section** button causes the cross section editor to pop up. This editor is shown in Figure 3.5. As shown, each cross section has a **Reach** name, **River Station**, and a **Description**. The **Reach** and **River Station** identifiers are used to describe where the cross section is located in the river system. The "River Station" identifier does not have to be the actual river station (miles or kilometers) at which the cross section is located on the stream, but it does have to be a numeric value (e.g. 1.1, 2, 3.5, etc.). The numeric value is used to place cross

sections in the appropriate order within a reach. Cross sections are ordered within a reach from the highest river station upstream to the lowest river station downstream.

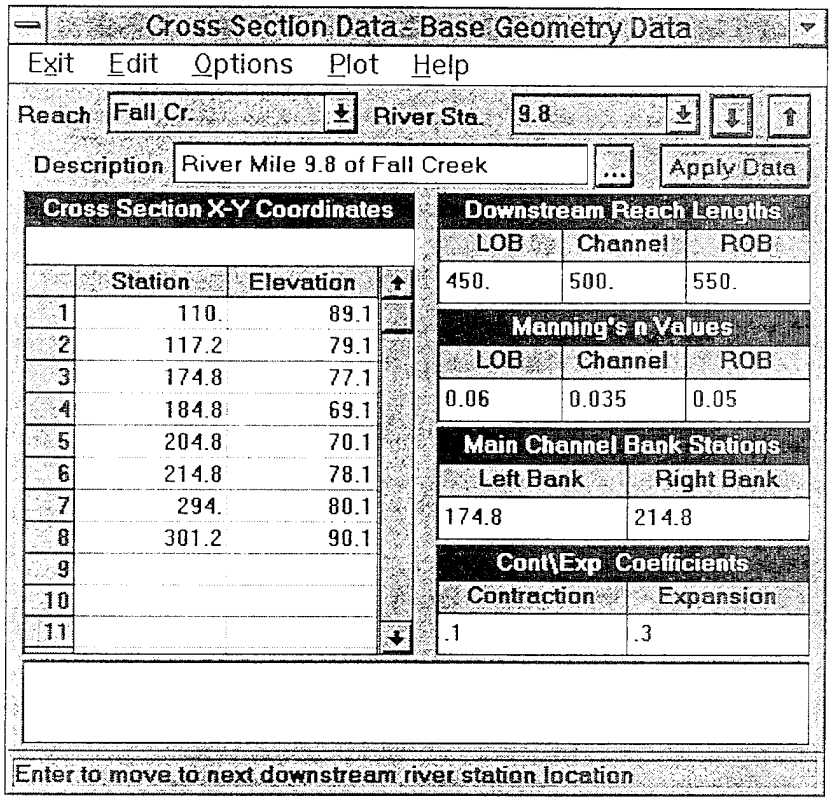


Figure 3.5 Cross Section Data editor

The basic data required for each cross section are shown on the Cross Section Data editor in Figure 3.5. Additional cross section features are available under **Options** from the menu bar. These options include: adding, copying, renaming and deleting cross sections; adjusting cross section elevations, stations, and n-values; ineffective flow areas; levees; blocked obstructions; adding a lid to a cross section; horizontal variation of n-values; and setting the maximum number of station and elevation points.

Also available from the Cross Section Data editor is the ability to plot any cross section or reach profile. **Edit** features are available to cut, copy, paste, insert, and delete data from the Cross Section X-Y Coordinates grid.

Once the cross-section data are entered, the modeler can then add any hydraulic structures such as bridges, culverts, weirs and spillways. Data editors, similar to the cross section data editor, are available for the various types of hydraulic structures. If there are any stream junctions in the river system, additional data are required for each junction. The Junction data editor is available from the Geometric Data window.

Once geometric data are entered, the data should be saved to a file on the hard disk. This is accomplished by selecting the **Save Geometric Data As** option from the **File** menu on the Geometric Data editor. This option allows the user to enter a title for the geometric data. A filename is automatically established for the geometric data, and then saved to the disk. Once a title is established, geometric data can be saved periodically by selecting **Save Geometric Data** from the File menu of the Geometric Data editor.

Entering Flow and/or Sediment Data

Once the geometric data are entered, the modeler can then enter any flow and/or sediment data that are required. The type of data that are entered will depend on the type of analysis that is going to be performed. If the modeler is performing a steady flow analysis, then Steady Flow Data must be entered. If an unsteady flow analysis is going to be performed, then Unsteady Flow Data must be entered. Likewise, if the modeler is going to perform sediment transport calculations, then Sediment Data must be entered. Data entry forms for these various types of data are available under the **Edit** menu bar option on the HEC-RAS main window.

An example of the flow data entry form is shown in Figure 3.6, which is the **Steady Flow Data** form. As shown in Figure 3.6, Steady Flow Data consist of: the number of profiles to be computed; the flow data; and the river system boundary conditions. At least one flow must be entered for every reach within the system. Additionally, flow can be changed at any location within the river system. Flow values must be entered for all profiles.

Boundary conditions are required in order to perform the calculations. If a subcritical flow analysis is going to be performed, then only the downstream boundary conditions are required. If a supercritical flow analysis is going to be performed, then only the upstream boundary conditions are required. If the modeler is going to perform a mixed flow regime calculation, then both upstream and downstream boundary conditions are required. The Boundary Conditions data entry form can be brought up by pressing the **Enter Boundary Conditions** button from the Steady Flow Data entry form.

Once all of the steady flow data and boundary conditions are entered, the modeler should save the data to the hard disk. This can be accomplished by selecting **Save Flow Data As** from the **File** option on the Steady Flow Data menu bar. Flow data is saved in a separate file. The user is only required to enter a title for the data, the filename is automatically assigned.

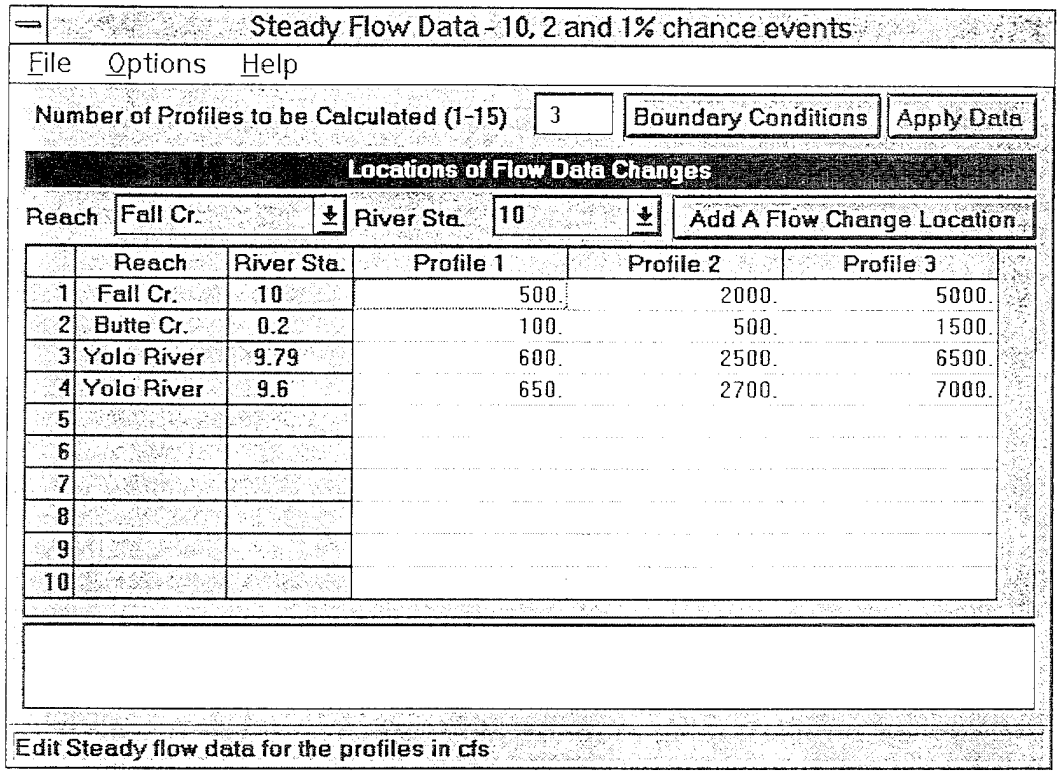


Figure 3.6 Steady Flow Data window

Performing The Hydraulic Computations

Once all of the geometric data and flow data are entered, the modeler can begin to perform the hydraulic calculations. As stated previously, there are four types of calculations that can be performed: Steady Flow Analysis; Unsteady Flow Analysis; Sediment Transport Analysis; and Hydraulic Design Functions. The modeler can select any of the available hydraulic analyses from the **Simulate** menu bar option on the HEC-RAS main window. An example simulation window is shown in Figure 3.7, which is the Steady Flow Analysis window.

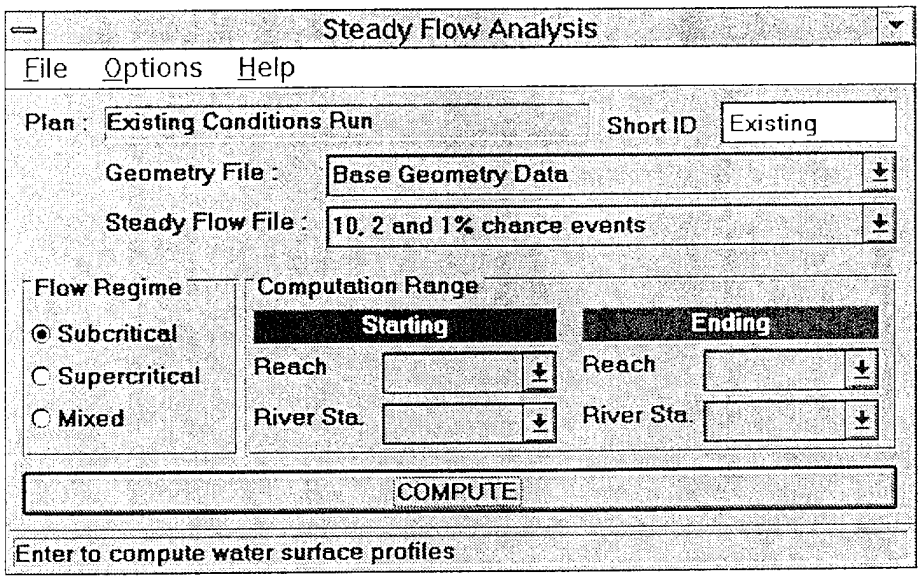


Figure 3.7 Steady Flow Analysis window

As shown in Figure 3.7, the modeler puts together a **Plan** by selecting a specific set of geometric data and flow data. A Plan can be put together by selecting **New Plan** from the **File** menu bar option of the Steady Flow Analysis window. Once a plan has been put together, the modeler must select a **Flow Regime** for which the model will perform calculations. Subcritical, Supercritical, or Mixed flow regime calculations are available.

An option for selecting a **Computation Range** is also available. This option allows the modeler to select a specific piece of the river system to work on. By selecting this option, the system will only perform calculations on the smaller subset of the full data. This option is useful when it is desired to calibrate the model or adjust data for a small piece of the river system. This option should be left blank when it is desired to perform calculations for the full river system. Also, once the modeler is finished working on the smaller subset of the river system, the model should always be re-run for the entire system. If the model is not re-run for the entire system, output will only be available for the smaller subset.

Additional features are available under the **Options** menu for: setting output options; conveyance calculation options; friction slope methods; calculation tolerances; critical depth computation method; data checking; setting log file levels; and viewing the log file output.

Once the modeler has selected a Plan and set all of the calculation options, the steady flow calculations can be performed by pressing the **Compute** button at the bottom of the Steady Flow Analysis window. When this button is pressed, the HEC-RAS system packages up all the data for the selected plan and writes

it to a run file. The system then runs the steady flow model (SNET) and passes it the name of the run file. This process is executed in a separate window. Therefore, the modeler can work on other tasks while it is executing.

Viewing and Printing Results

Once the model has finished all of the computations, the modeler can begin viewing the results. Several output features are available under the **View** option from the main window. These options include: cross section plots; profile plots; rating curve plots; X-Y-Z perspective plots; tabular output at specific locations; tabular output for many locations; and the summary of errors, warnings, and notes.

An example of a cross section plot is shown in Figure 3.8. The user can plot any cross section by simply selecting the appropriate reach and river station from the list boxes at the top of the plot. The user can also step through the plots by using the up and down arrow buttons. Several plotting features are available under the **Options** menu of the Cross Section plot. These options include: zoom in; zoom out; selecting which plans, profiles and variables to plot; and control over the lines, symbols, labels, scaling, and grid options.

Hard copy outputs of the graphics can be accomplished in two different ways. Plots can be sent directly from HEC-RAS to whichever printer or plotter the user has defined under the Windows Print Manager. Plots can also be sent to the Windows clipboard. Once the plot is in the clipboard it can then be pasted into other programs, such as a word processor. Both of these options are available from the **File** menu on the various plot windows.

An example of a profile plot is shown in Figure 3.9. All of the options available in the cross section plot are also available in the profile plot. Additionally, the user can select which specific reaches to plot when a multiple-reach river system is being modeled.

An example of an X-Y-Z Perspective Plot is shown in Figure 3.10. The user has the option of defining the starting and ending location for the extent of the plot. The plot can be rotated left or right, and up or down, in order to get different perspectives of the river reach. The computed water surface profiles can be overlaid on top of the cross section data. The graphic can be sent to the printer or plotter directly, or the plot can be sent through the Windows Clipboard to other programs.

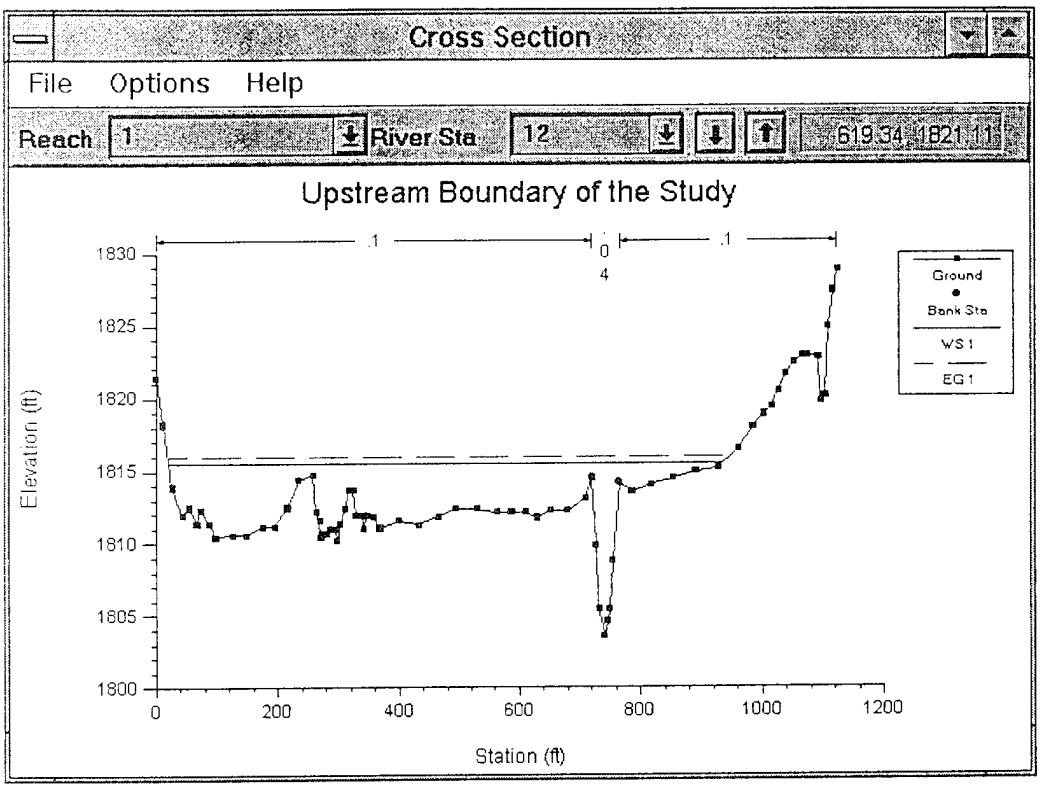


Figure 3.8 Cross Section Plot

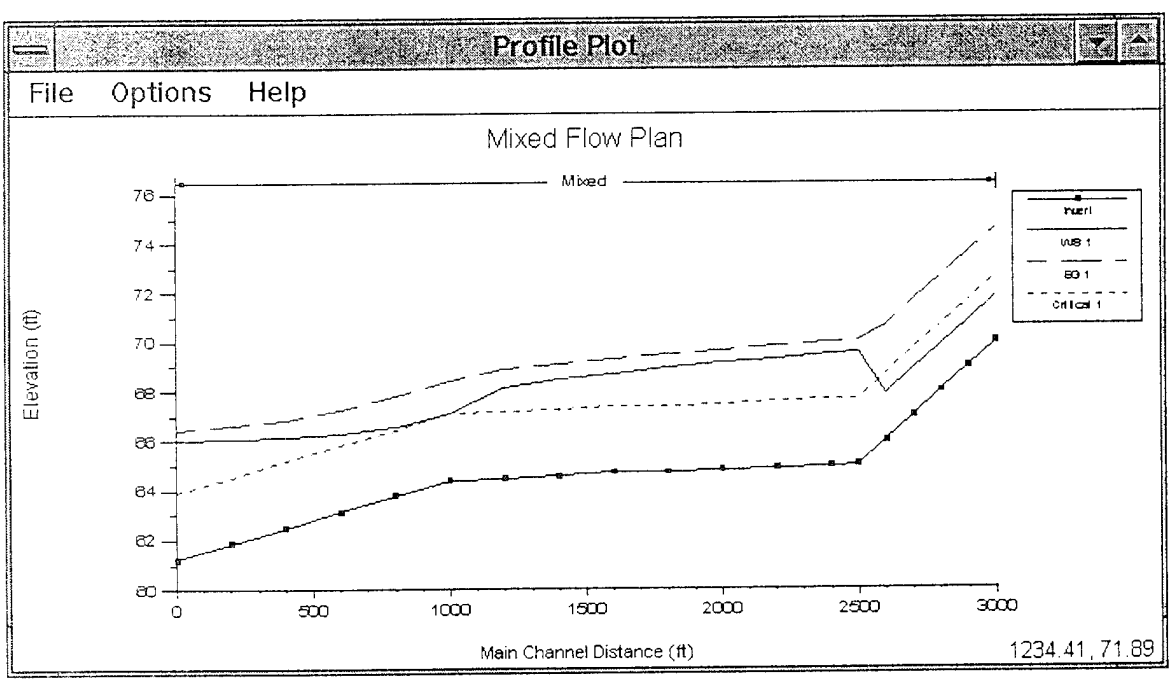


Figure 3.9 Profile Plot

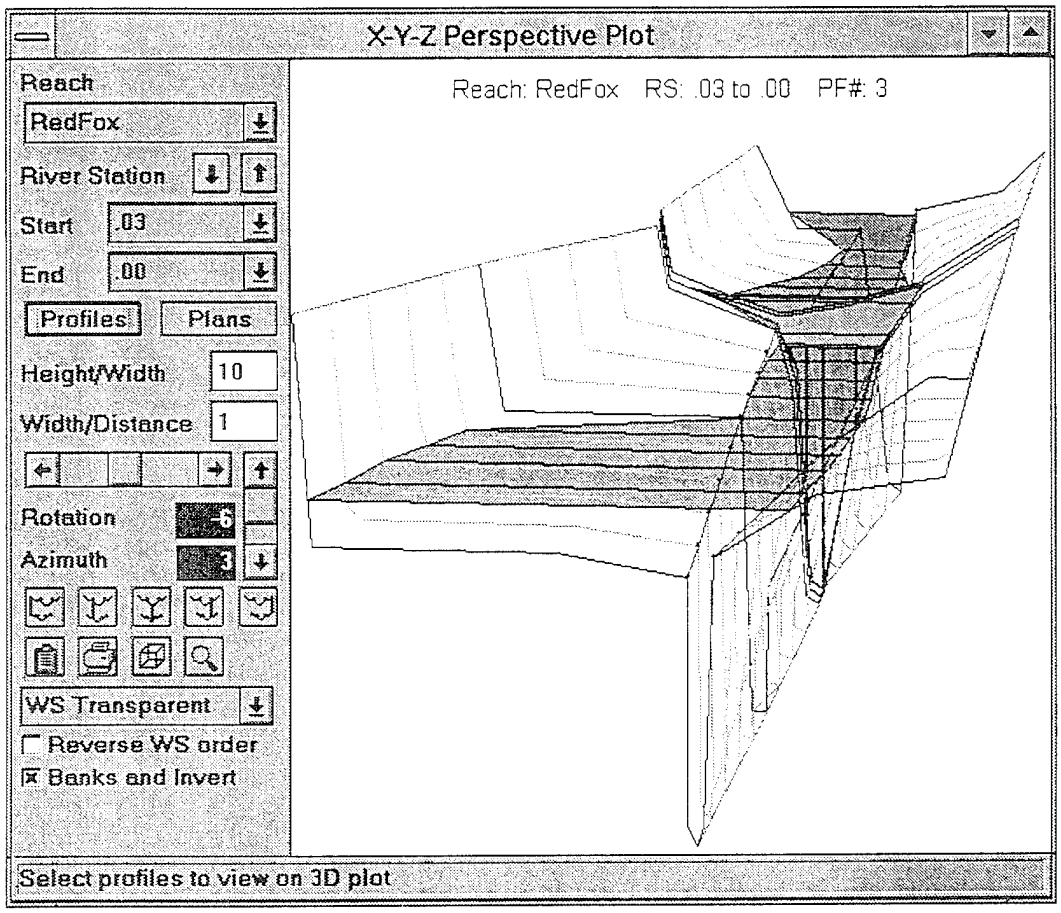


Figure 3.10 X-Y-Z Perspective Plot of River Reach with a Bridge

Tabular output is available in two different formats. The first type of tabular output provides detailed hydraulic results at a specific cross section location (cross section table). An example of this type of tabular output is shown in Figure 3.11.

Cross Section Output

File Type Options

Reach RedFox River Sta. 01 Profile 5

HEC-RAS	Plan: RedFox	Reach: RedFox	Riv Sta: 01	Profile: 5	
W.S. Elev (ft)	18.38	Element	Left O.B.	Channel	Right O.B.
Vel Head (ft)	1.23	Wt. n-Val.	0.050	0.030	0.100
E.G. Elev (ft)	19.61	Reach Len. (ft)	500.00	500.00	500.00
E.G. Slope (ft/ft)	0.001834	Flow Area (sq ft)	850.01	744.94	213.55
Q Total (cfs)	10000.00	Flow (cfs)	2387.29	7541.08	71.63
Top Width (ft)	882.18	Top Width (ft)	259.17	65.00	558.02
Vel Total (ft/s)	5.53	Avg. Vel. (ft/s)	2.81	10.12	0.34
Max Chl Dpth (ft)	14.38	Hydr. Depth (ft)	3.28	11.46	0.38
Crit W.S. (ft)		Wetted Per. (ft)	259.33	71.46	558.04
Conv. Total (cfs)	233483.5	Conv. (cfs)	55739.2	176071.7	1672.6

Errors, Warnings and Notes

Warning - The velocity head has changed by more than 0.5 ft (0.15 m). This may indicate the need for additional cross sections.
Warning - The conveyance ratio (upstream conveyance divided by downstream

Calculated water surface from energy equation.

Figure 3.11 Tabular Cross Section Output

The second type of tabular output shows a limited number of hydraulic variables for several cross sections and multiple profiles. An example of this type of tabular output is shown in Figure 3.12. There are several standard tables that are pre-defined and provided to the user under the **Tables** menu from the profile output tables. Users can also define their own tables by specifying what variables they would like to have in a table. User specified table headings can be saved and then selected later as one of the standard tables available to the project.

Tabular output can be sent directly to the printer or passed through the clipboard in the same manner as the graphical output described previously. This option is also available under the **File** menu on each of the table forms.

Profile Output Table - Standard Table 1

File Options Std. Tables User Tables Help

HEC-RAS Plan: RedFox Reach: RedFox

River Sta.	Q Total	Min Ch El	W.S. Elev	Crit W.S.	E.G. Elev	E.G. Slope	Vel Chnl	Flow Area
	(cfs)	(ft)	(ft)	(ft)	(ft)	(ft/ft)	(ft/s)	(sq ft)
.03	3000.00	9.50	15.08	15.08	16.99	0.012633	11.08	270.80
.03	4500.00	9.50	16.40	16.40	18.67	0.011823	12.09	372.22
.03	6500.00	9.50	17.81	17.81	20.47	0.011238	13.10	496.17
.03	9000.00	9.50	19.28	19.28	22.32	0.010672	14.01	642.41
.03	10000.00	9.50	19.79	19.79	22.98	0.010548	14.34	697.32
.02	3000.00	7.10	13.28		14.06	0.003444	7.08	423.77
.02	4500.00	7.10	15.25		15.96	0.002289	6.75	666.74
.02	6500.00	7.10	16.76		17.58	0.002261	7.28	892.29
.02	9000.00	7.10	18.84		19.61	0.001868	7.03	1280.18
.02	10000.00	7.10	19.62		20.35	0.001632	6.90	1463.09
.01	3000.00	4.00	10.36		12.20	0.005764	10.89	276.70
.01	4500.00	4.00	12.19		14.45	0.005429	12.09	372.25
.01	6500.00	4.00	14.66		16.41	0.003248	11.23	761.29
.01	9000.00	4.00	17.42		18.76	0.002114	10.36	1315.00
.01	10000.00	4.00	18.38		19.61	0.001834	10.12	1808.50

Average velocity of flow in main channel.

Figure 3.12 Profile Output Table

Importing HEC-2 Data

An important feature of HEC-RAS is the ability to import HEC-2 data. This feature makes it easy for a user to import existing HEC-2 data sets and start using HEC-RAS immediately. To import HEC-2 data, do the following:

1. Start a new project by selecting **New Project** under the **File** menu option on the HEC-RAS main window (Figure 3.13).

2. Select the **Import HEC-2 Data** option under the **File** menu on the main window (Figure 3.13). A popup window will appear (Figure 3.14), which will allow you to select a drive, path, and filename for the HEC-2 data file. Once you have selected the file, press the **OK** button.

The data are automatically saved in HEC-RAS format with default names and titles. The user can change the titles at any time by using the **Rename** feature, which is available from the **File** menu of the various data editors.

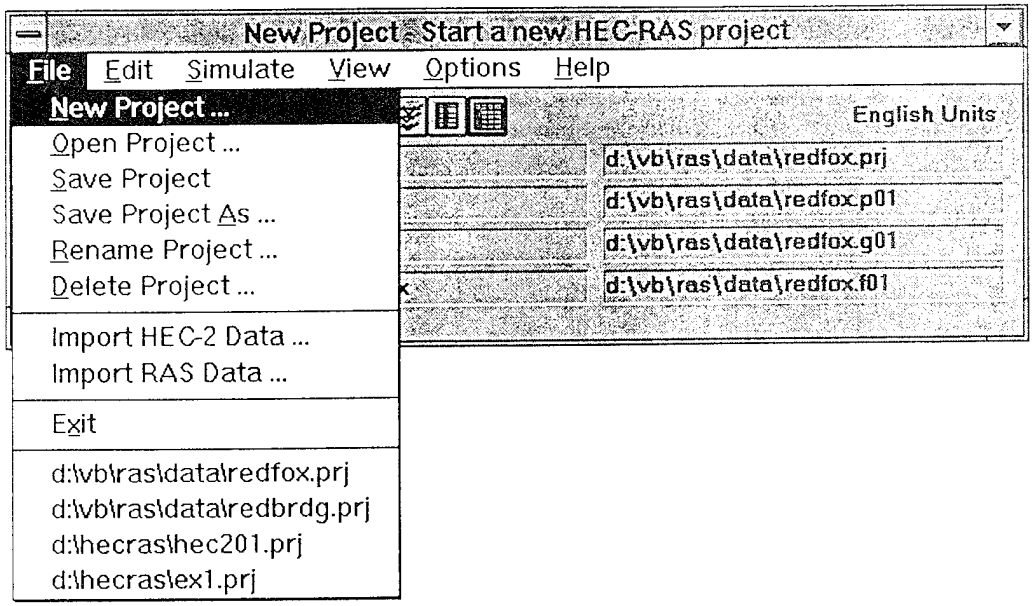


Figure 3.13 HEC-RAS Main Window With File Menu Options Shown

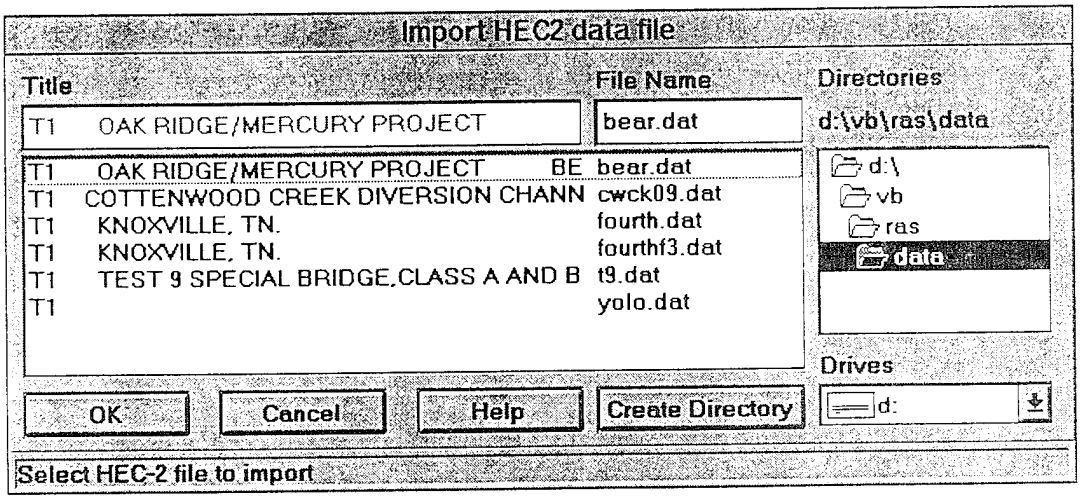


Figure 3.14 PopUp Window For Importing HEC-2 Data

The data associated with HEC-2 options that are not currently supported in HEC-RAS are ignored. This includes: vertical n values; k values; cross section interpolation; split flow data; channel improvements; n value calibration; and ice options. This information will be added to the HEC-2 import routine as the options become available in HEC-RAS.

When bridge data are imported, the user must take special care to ensure that the data are correctly representing the bridge. The bridge routines in HEC-RAS are different than HEC-2, and therefore you may have to modify some data and/or enter some additional data. Whenever you import an HEC-2 data set with bridge data, carefully review all the data for each bridge. Chapter 6 of this user's manual describes the required data for bridges in HEC-RAS. Some key differences between HEC-2 and HEC-RAS are as follows:

1. **Special Bridge Data Sets**

   HEC-RAS does not use a trapezoidal approximation for low flow through the bridge opening. The actual bridge opening is used in both the Yarnell method and the momentum method.. This could be a problem for HEC-2 special bridge data sets that do not include low chord information on the BT data. If you have a data set like this, you will need to modify the bridge deck information after the data have been imported.

   The pressure flow equations use the actual bridge opening, defined by the ground and the bridge data. In HEC-2, the user was required to enter an area for pressure flow. If the actual bridge opening produces a different area than what the user had entered in the HEC-2 data deck, the program will get different results for pressure flow, and pressure and weir flow answers.

   Pier information from the SB record is incorporated as a single pier in the HEC-RAS data set. Piers are treated as a separate pieces of data in HEC-RAS. For special bridges that have piers, you may want to change the single pier to multiple piers, depending on what is actually at the bridge. Pier information can be modified using the **Pier** editor.

2. **Normal Bridge Data Sets**

   Because piers are treated as a separate piece of data in HEC-RAS, they must not be included in the cross section data or the bridge deck. Since it is common to include pier information as part of the cross section or bridge deck in HEC-2, these data will need to be modified. For data sets that have piers, you will need to remove the pier information from the cross section or bridge deck, and then add the information back in using the **Pier** editor.

Getting and Using Help

At this time, on-line help is not available in HEC-RAS. The Help system will be added before the first official release. For any questions that you currently have, please refer to this user's manual or the hydraulics reference manual.

CHAPTER 4

Example Application

This chapter provides an example application of how to perform steady flow water surface profile calculations with HEC-RAS. The user is taken through a step by step procedure of how to enter data, perform calculations, and view the results.

In order to get the most out of this chapter, you should perform each of the steps on your own computer. Also, before you try the example application, you should have read the first three chapters in this manual.

Contents

- Starting a New Project
- Entering Geometric Data
- Entering Steady Flow Data
- Performing the Hydraulic Calculations
- Viewing Results
- Printing Graphics and Tables
- Exiting the Program

Starting a New Project

To begin this example, let's first start the HEC-RAS program. Double click the HEC-RAS icon in Windows. The main window should appear as shown in Figure 4.1 (except yours will be blank the first time you start the program).

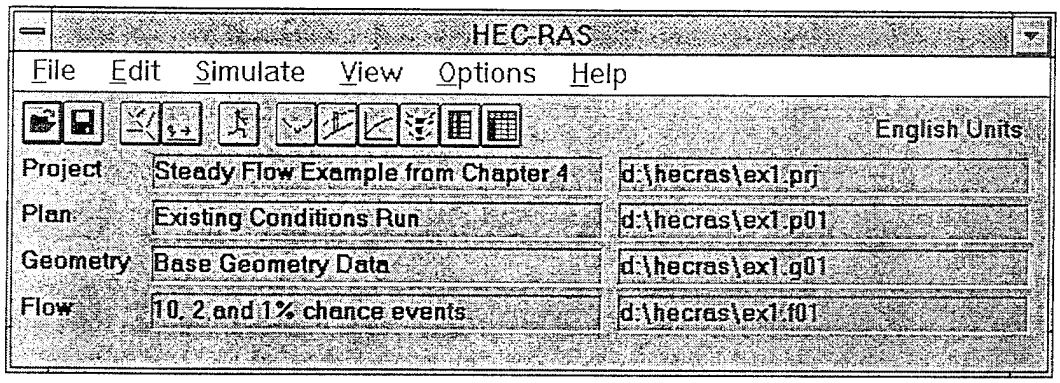


Figure 4.1 HEC-RAS Main Window

The first step in developing an HEC-RAS application is to start a new project. Go to the **File** menu on the main window and select **New Project.** The New Project window should appear as shown in Figure 4.2 (except the title and file name fields will be blank when it first comes up).

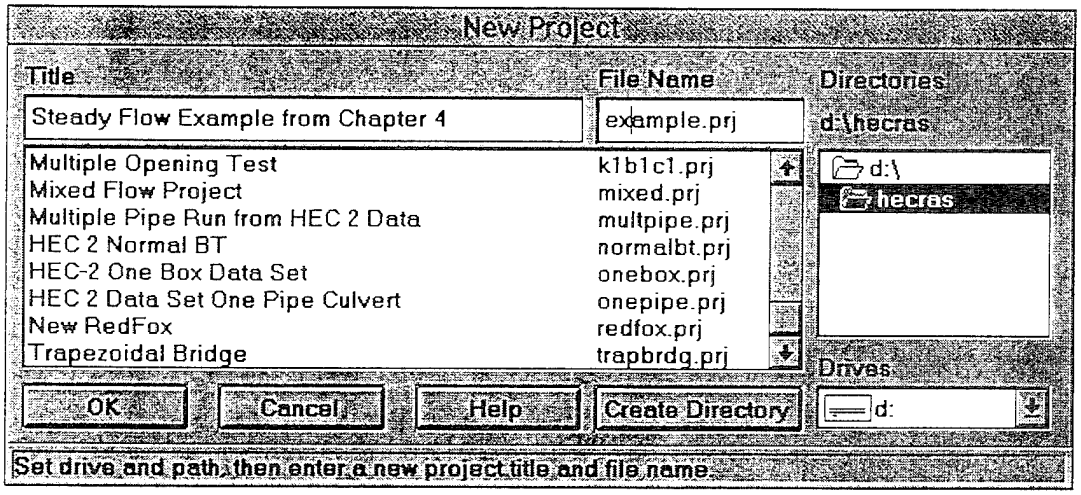


Figure 4.2 New Project Window

First set the drive (e.g. C:) and the directory that you would like to work in. Next enter the project title and filename as shown in Figure 4.2. Once you have entered the information, press the **OK** button to have the data accepted.

Entering Geometric Data

The next step in developing a steady flow model with HEC-RAS is to enter the geometric data. This is accomplished by selecting **Geometric Data** from the **Edit** menu on the HEC-RAS main window. Once this option is selected the geometric data window will appear (Figure 4.3).

Drawing the Schematic of the River System

In this example we are going to develop a three-reach system as shown in Figure 4.3. Draw in the river system schematic by performing the following steps:

1. Click the **River Reach** button on the geometric data window.

2. Move the mouse pointer over to the drawing area and place the pointer at the location in which you would like to start drawing the first reach.

3. Hold down the left mouse button and draw the reach from upstream to downstream (in the positive flow direction).

4. The interface will prompt you to enter an identifier for the reach. This identifier is limited to 12 characters.

5. Repeat steps 1 through 4 for each reach. After you enter the identifier for reach three you will also be prompted to enter an identifier for the junction (location where two or more streams join or split apart).

Once you have finished drawing in the river system, there are several options available for editing the schematic. These options include: deleting reaches, changing labels, and moving any objects (objects are labels, junctions, and the ends of reaches). The editing features are located under the **Edit** menu on the Geometric Data window. **Note: when you first draw your schematic there will not be any tic marks representing cross sections as shown in Figure 4.3. The tic marks only show up after you have entered cross section data.**

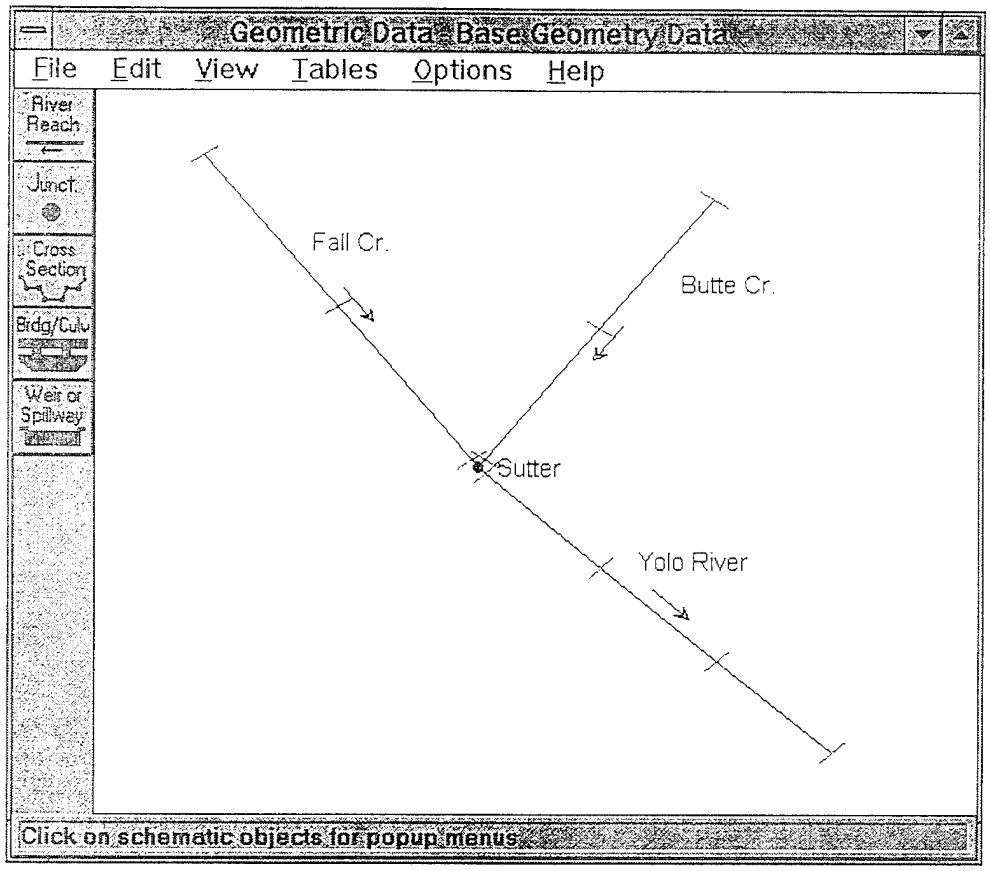


Figure 4.3 **Geometric Data Window with example river schematic**

Entering Cross Section Data

The next step is to enter the cross section data. This is accomplished by pressing the **Cross Section** button on the Geometric Data window (Figure 4.3). Once this button is pressed, the Cross Section Data editor will appear as shown in Figure 4.4 (except yours should be blank). To enter cross section data do the following:

1. Select a **Reach** to work with. For this example start with the Fall Cr. reach.

2. Go to the **Options** menu and select **Add a new Cross Section**. An input box will appear prompting you to enter a river station identifier for the new cross section. The identifier does not have to be the actual river station, but it must be a numeric value. The numeric value describes where this cross section is located in reference to all the other cross sections within the reach. Cross sections are located from upstream (highest river station) to downstream (lowest river station). For this cross section enter a value of 10.0.

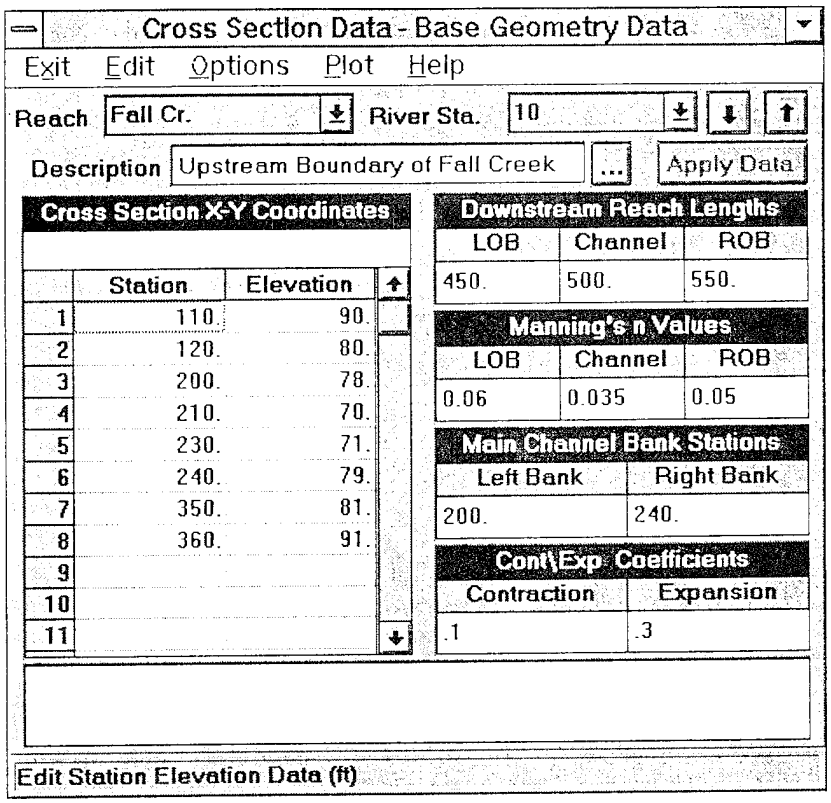


Figure 4.4 Cross Section Data Editor with example data

3. Enter all of the data for this cross section as it is shown in Figure 4.4.

4. Once all the data are entered press the **Apply Data** button. This button is used to tell the interface that you want the data to be accepted into memory. This button does not save the data to your hard disk, that can only be accomplished from the **File** menu on the Geometric Data window.

5. Plot the cross section to visually inspect the data. This is accomplished by pressing the **Plot Cross Section** option under the **Plot** menu on the Cross Section Data Editor. The cross section should look the same as that shown in Figure 4.5.

In general, the five steps listed would be repeated for every cross section that is entered. In order to reduce the amount of data entry for this example, the current cross section will be copied and adjusted to represent other cross sections within the river system.

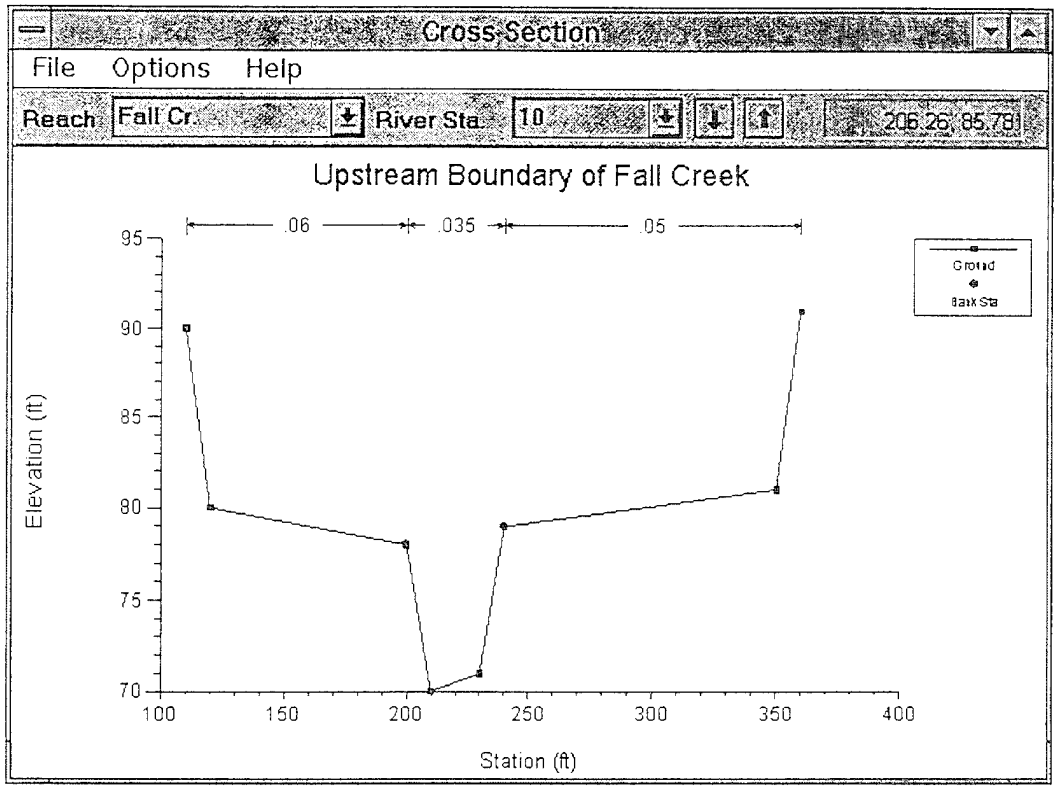


Figure 4.5 Cross Section Plot for river mile 10.0 of Fall Creek

The following steps should be followed to copy the current cross section:

1. Go to the **Options** menu on the Cross Section Data Editor and select **Copy Current Cross Section**. An input box will appear prompting you to select a reach and enter a river station for the new cross section. For this example, keep the reach as Fall Cr. and enter a new river station of 9.9. Press the **OK** button and the new cross section will appear in the editor.

2. Change the description for the cross section to "River Mile 9.9 of Fall Creek."

3. Adjust all the elevations of the cross section by -0.5 feet. This is accomplished by selecting the **Adjust Elevations** feature from the **Options** menu on the Cross Section Data Editor.

4. Adjust the cross section stationing to reduce the overbanks by 10%. This is accomplished by selecting the **Adjust Stations** feature from the **Options** menu on the Cross Section Data Editor, then select **Multiply by a Factor**. When the input box appears for this option,

three data entry fields will be available to adjust the stationing of the left overbank, channel, and the right overbank separately. Enter values of 0.90 for the right and left overbanks, but leave the main channel field blank. This will reduce the stationing of both overbanks by 10%, but the main channel will not be changed.

5. Downstream reach lengths remain the same for this cross section.

6. Press the **Apply Data** button.

7. Plot the cross section to visually inspect it.

These seven steps should be repeated to enter all the data for Fall Creek and the Yolo River. The necessary adjustments are listed in Table 4.1. Perform the cross section duplications in the order that they are listed in the table. Make sure to change the description of each cross section, and also press the **Apply Data** button after making the adjustments for each cross section.

Table 4.1 Cross Section adjustments for duplicating sections

Cross Section		Adjusted Elevation	Adjusted Stationing			Downstream Reach Lengths		
Reach	River Sta.		Left O.B.	Channel	Right O.B.	Left O.B.	Channel	Right O.B.
Fall Cr.	9.8	-0.4	0.80	-	0.80	0.0	0.0	0.0
Yolo Ri.	9.79	-0.1	1.20	1.20	1.20	500	500	500
Yolo Ri.	9.7	-0.5	1.20	1.20	1.20	500	500	500
Yolo Ri.	9.6	-0.3	-	-	-	500	500	500
Yolo Ri	9.5	-0.2	-	-	-	0.0	0.0	0.0

This completes all the cross section data for Fall Creek and the Yolo River. Now let's work on entering the data for the Butte Creek tributary. To enter the first cross section in the Butte Creek tributary do the following:

1. Go to the **Reach** text box on the Cross Section Data Editor and select the **Butte Cr.** reach.

2. Select **Add a new Cross Section** from the **Options** menu. When the popup box appears prompting you to enter a new river station, enter a value of **0.2**.

3. Enter all the data for this cross section as shown in Figure 4.6.

4. Once all the data are entered for this section, press the **Apply Data** button.

5. Plot the cross section to inspect the data.

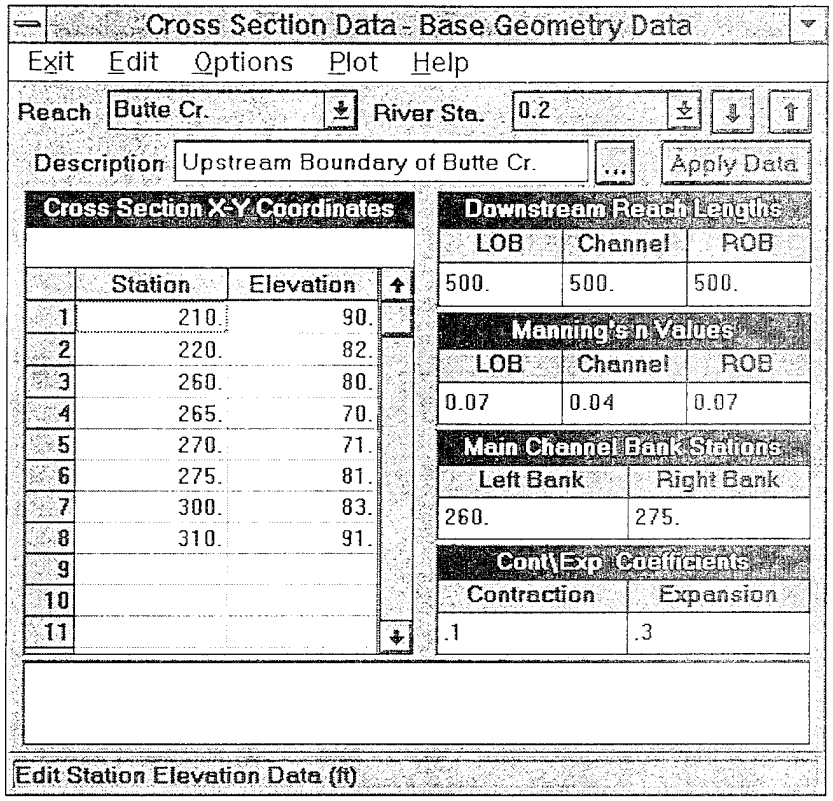


Figure 4.6 Cross Section Editor with river mile 0.2 of Butte Creek

There are two other cross sections that need to be developed for the Butte Creek tributary. These two cross sections will be developed by duplicating the cross section that you just entered, and then adjusting the elevations and stationing. The necessary adjustments are listed in Table 4.2. Perform the cross section adjustments in the order that they are listed in the table. Make sure to change the description of each cross section and press the **Apply Data** button after editing is complete.

Table 4.2 Cross Section adjustments for Butte Creek sections

Cross Section		Adjusted Elevation	Adjusted Stationing			Downstream Reach Lengths		
Reach	River Sta.		Left O.B.	Channel	Right O.B.	Left O.B.	Channel	Right O.B.
Butte Cr.	0.1	-0.6	-	-	-	500	500	500
Butte Cr.	0.0	-0.3	-	-	-	0.0	0.0	0.0

Now that all of the cross section data are entered, save the data to a file before continuing. Saving the data to a file is accomplished by selecting the "**Save Geometry Data As**" option from the **File** menu on the Geometric Data window. After selecting this option you will be prompted to enter a description of the geometric data. Enter "Base Geometry Data" for this example, then press the **OK** button. A file name is automatically assigned to the geometry data based on what you entered for the project filename.

Entering Junction Data

The next step is to enter the junction data. Junction data consist of a description, and reach lengths across the junction. In this example there is only one junction, which is labeled **Sutter**. Junction data is entered by pressing the **Junction** button on the Geometric Data window. Enter the junction data as shown in Figure 4.7.

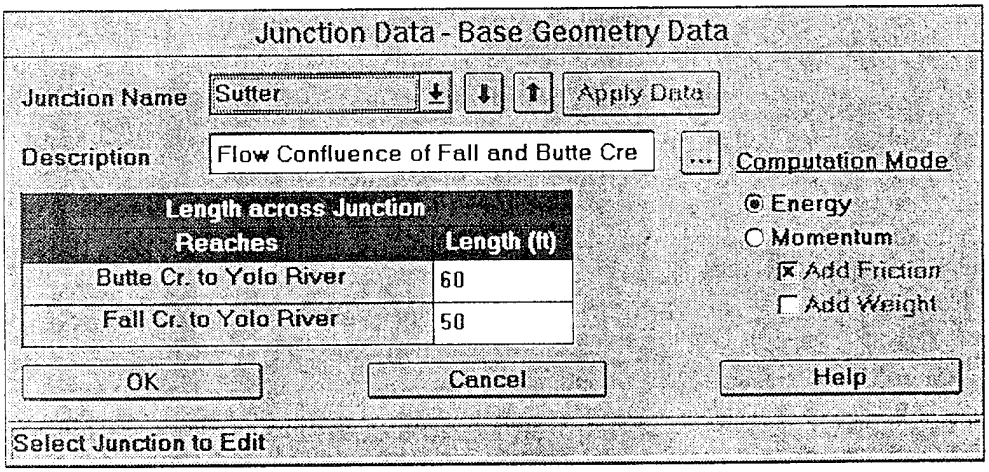


Figure 4.7 Junction Data Editor, with Sutter junction data

Reach lengths across the junction are entered in the junction editor, rather than in the cross section data. This allows for the lengths across very complicated confluences (i.e. flow splits) to be accommodated. In the cross section data, the reach lengths for the last cross section of each reach should be left blank or set to zero.

In this example the energy equation will be used to compute the water surface profile through the junction. If the momentum equation is selected, then an angle must be entered for one or more of the reaches flowing into or out of a junction. The momentum equation is set up to account for the angle of the flow entering the junction.

Once you have all of the data entered for the junction, apply the data and close the window by pressing the **OK** button.

Saving The Geometry Data

At this point in the example, all of the geometric data has been entered. Before we continue with the example, you should save the geometric data to the hard disk. Since the data have already been saved once, you simply have to select **Save Geometry Data** from the **File** menu on the Geometric Data window. We can now go on to enter the Steady Flow data.

Entering Steady Flow Data

The next step in developing the required data to perform steady flow water surface profile calculations is to enter the steady flow data. To bring up the steady flow data editor, select **Steady Flow Data** from the **Edit** menu on the HEC-RAS main window. The Steady Flow Data editor should appear as shown in Figure 4.8.

The first piece of data to enter is the number of profiles to be calculated. For this example enter "3" as shown in Figure 4.8. The next step is to enter the flow data. Flow data are entered from upstream to downstream for each reach. At least one flow rate must be entered for every reach in the river system. Once a flow value is entered at the upstream end of a reach, it is assumed that the flow remains constant until another flow value is encountered within the reach. Additional flow values can be entered at any cross section location within a reach.

Steady Flow Data - 10, 2 and 1% chance events

File Options Help

Number of Profiles to be Calculated (1-15) 3 Boundary Conditions Apply Data

Locations of Flow Data Changes

Reach Butte Cr. River Sta. 0.2 Add A Flow Change Location

	Reach	River Sta.	Profile 1	Profile 2	Profile 3
1	Fall Cr.	10	500.	2000.	5000.
2	Butte Cr.	0.2	100.	500.	1500.
3	Yolo River	9.79	600.	2500.	6500.
4	Yolo River	9.6	650.	2700.	7000.
5					
6					
7					
8					
9					
10					

Edit Steady flow data for the profiles in cfs

Figure 4.8 Steady Flow Data Editor, with example problem data

In this example, flow data will be entered at the upstream end of each reach. An additional flow change location will be entered at river mile 9.6 of the Yolo River. To add an additional flow change location into the table, first select the Yolo River from the **Reach** list box. Next select the desired river station location (9.6 in this example) from the **River Sta.** list box. Finally, press the **Add A Flow Change Location** button. The new flow location should appear in the table. Now enter all of the flow data into the table as shown in Figure 4.8.

The next step is to enter any boundary conditions that may be required. To enter boundary condition data, press the **Enter Boundary Conditions** button at the top of the Steady Flow Data editor. The boundary conditions editor should appear as shown in Figure 4.9.

Boundary conditions are necessary to establish the starting water surface at the ends of the river system. A starting water surface is necessary in order for the program to begin the calculations. In a subcritical flow regime, boundary conditions are only required at the downstream ends of the river system. If a supercritical flow regime is going to be calculated, boundary conditions are only necessary at the upstream ends of the river system. If a mixed flow regime calculation is going to be made, then boundary conditions must be entered at all ends of the river system.

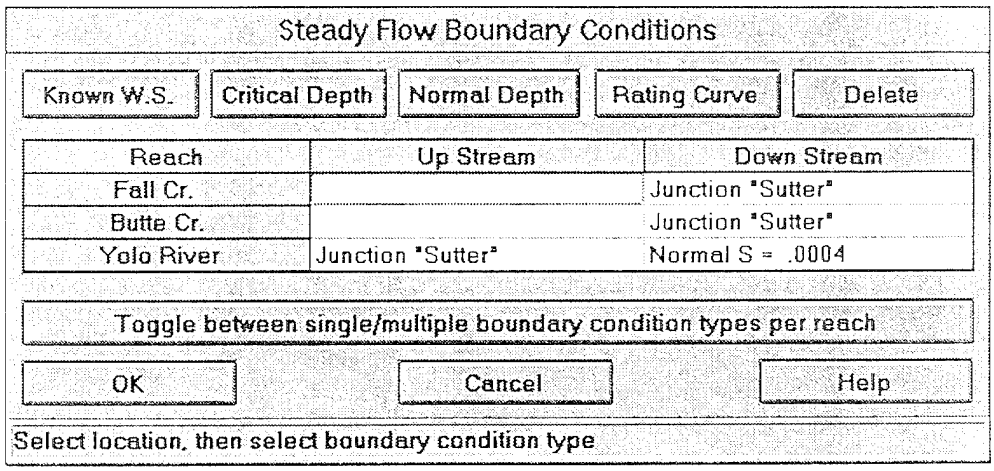


Figure 4.9 Steady Flow Boundary Conditions

The boundary conditions editor contains a table listing every reach. Each reach has an upstream and a downstream boundary condition. Connections to junctions are considered internal boundary conditions. Internal boundary conditions are automatically listed in the table, based on how the river system is connected in the geometric data editor. The user is only required to enter the necessary external boundary conditions.

In this example, it is assumed that the flow is subcritical throughout the river system. Therefore, it is only necessary to enter a boundary condition at the downstream end of the Yolo River. Boundary conditions are entered by first selecting the cell in which you wish to enter a boundary condition. Then the type of boundary condition is selected from the four available types listed above the table. The four types of boundary conditions consist of:

- Known water surface elevations
- Critical depth
- Normal depth
- Rating curve

For this example use the normal depth boundary condition. Once you have selected the cell for the downstream end of Yolo River, press the **Normal Depth** button. A popup box will appear requesting you to enter an average

energy slope at the downstream end of the Yolo River. Enter a value of 0.0004 (ft/ft), then press the **Enter** key. This completes all of the necessary boundary condition data. Press the **OK** button on the Boundary Conditions form to accept the data.

The last step in developing the steady flow data is to save the data to a file. To save the data, select the **Save Flow Data As** option from the **File** menu on the Steady Flow Data Editor. A popup box will prompt you to enter a description of the flow data. For this example enter "10, 2, and 1% chance events". Once the data are saved you can close the Steady Flow Data Editor.

Performing The Hydraulic Calculations

Now that all of the data have been entered, we can calculate the steady water surface profiles. To perform the simulations, go to the HEC-RAS main window and select **Steady Flow Analysis** from the **Simulate** menu. The Steady Flow Analysis window should appear as shown in Figure 4.10.

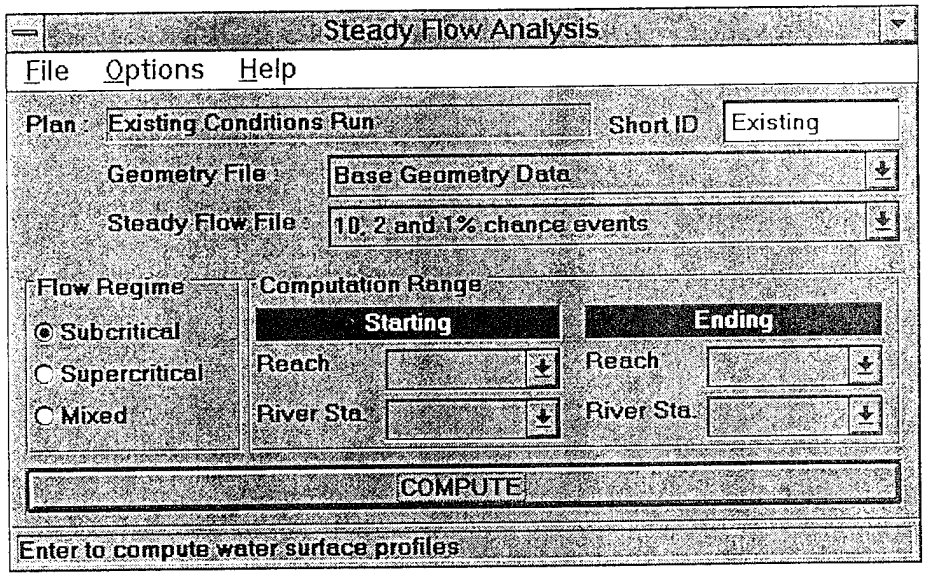


Figure 4.10 Steady Flow Analysis Simulation Window

The first step is to put together a **Plan**. The **Plan** defines which geometry and flow data are to be used, as well as providing a title and short identifier for the run. To establish a plan, select **New Plan** from the **File** menu on the Steady Flow Analysis window. Enter the plan title as "Existing Conditions Run" and then press the **OK** button. Enter a short identifier of "Existing" in the **Short ID** box.

The next step is to select the desired flow regime for which the model will perform calculations. For this example we will be performing **Subcritical** flow calculations only. Make sure that **Subcritical** is the selected flow regime. Additional job control features are available from the **Options** menu bar, but none are required for this example. Once you have defined a plan and set all the desired job control information, the plan information should be saved. Saving the plan information is accomplished by selecting **Save Plan** from the **File** menu of the Steady Flow Analysis window.

Now that everything has been set, the steady flow computations can be performed by pressing the **Compute** button at the bottom of the Steady Flow Simulation window. Once the compute button has been pressed, a separate window will appear showing you the progress of the computations. Once the computations have been completed, the computation window can be closed by double clicking the upper left corner of the window. At this time the Steady Flow Simulation window can also be closed.

Viewing Results

Once the model has finished all of the computations successfully, you can begin viewing the results. Several output options are available from the View menu bar on the HEC-RAS main window. These options include:

- Cross section plots
- Profile plots
- Rating curves
- X-Y-Z Perspective Plots
- Detailed tabular output at a specific cross section (cross section table)
- Limited tabular output at many cross sections (profile table)

Let's begin by plotting a cross section. Select **Cross Sections** from the **View** menu bar on the HEC-RAS main window. This will automatically bring up a plot of the first cross section in Fall Creek, as shown in Figure 4.11. Any cross section can be plotted by selecting the appropriate reach and river station from the list boxes at the top of the cross section plot window. The user can also step through the plots by using the up and down arrow buttons. Several plotting features are available from the **Options** menu bar on the cross section plot window. These options include: zoom in; zoom out; selecting which plans, profiles and variables to plot; and control over lines, symbols, labels, scaling, and grid options.

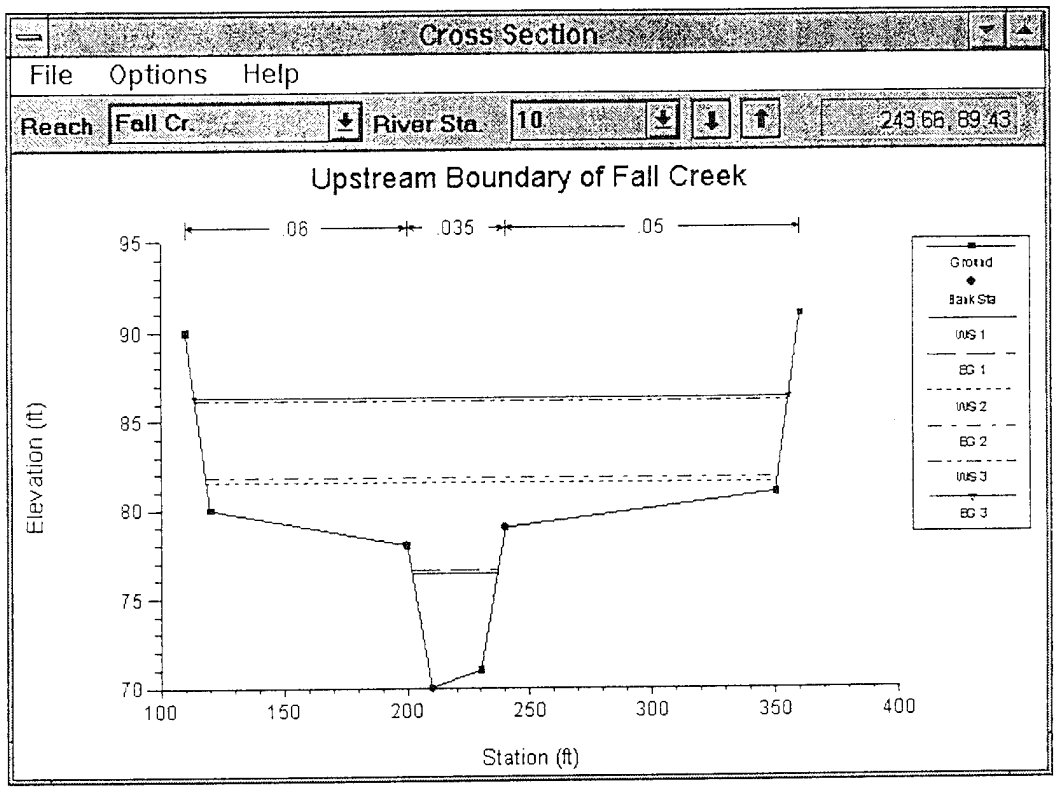


Figure 4.11 Cross Section Plot for Example Application

Select different cross sections to plot and practice using some of the features available under the **Options** menu bar.

Next let's plot a water surface profile. Select **Water Surface Profiles** from the **View** menu bar on the HEC-RAS main window. This will automatically bring up a water surface profile plot for the first reach, which is Yolo Creek in our example. To plot more than one reach, select **Reaches** from the **Options** menu bar on the profile plot. This option brings up a list of available reaches from which to choose. Select the Fall Cr. and Yolo River reaches. This should give you a profile plot as shown in Figure 4.12. Plot the additional profiles that were computed and practice using the other features available under the **Options** menu bar on the profile plot.

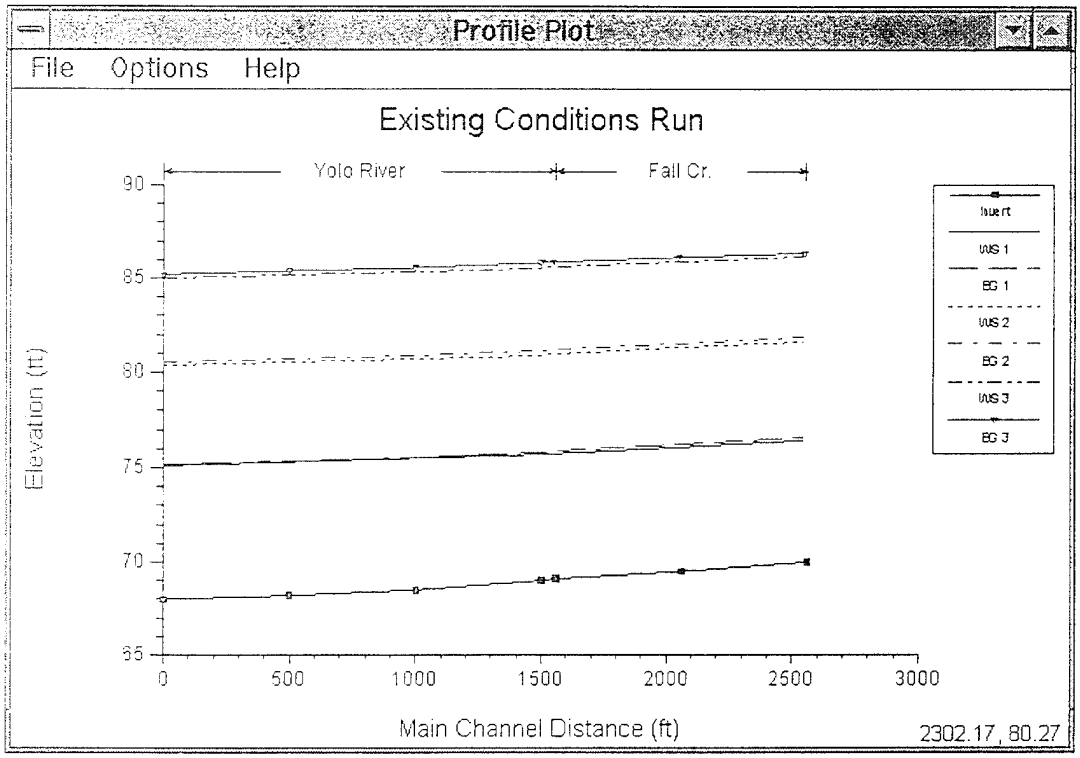


Figure 4.12 Profile Plot for Example Application

Now let's plot a computed rating curve. Select **Rating Curves** from the View menu on the HEC-RAS main window. A rating curve based on the computed water surface profiles will appear for the first cross section in Fall Creek, as shown in Figure 4.13. You can look at the computed rating curve for any location by selecting the appropriate reach and river station from the list boxes at the top of the plot. Plotting options similar to the cross section and profile plots are available for the rating curve plots. Plot rating curves for various locations and practice using the available plotting options.

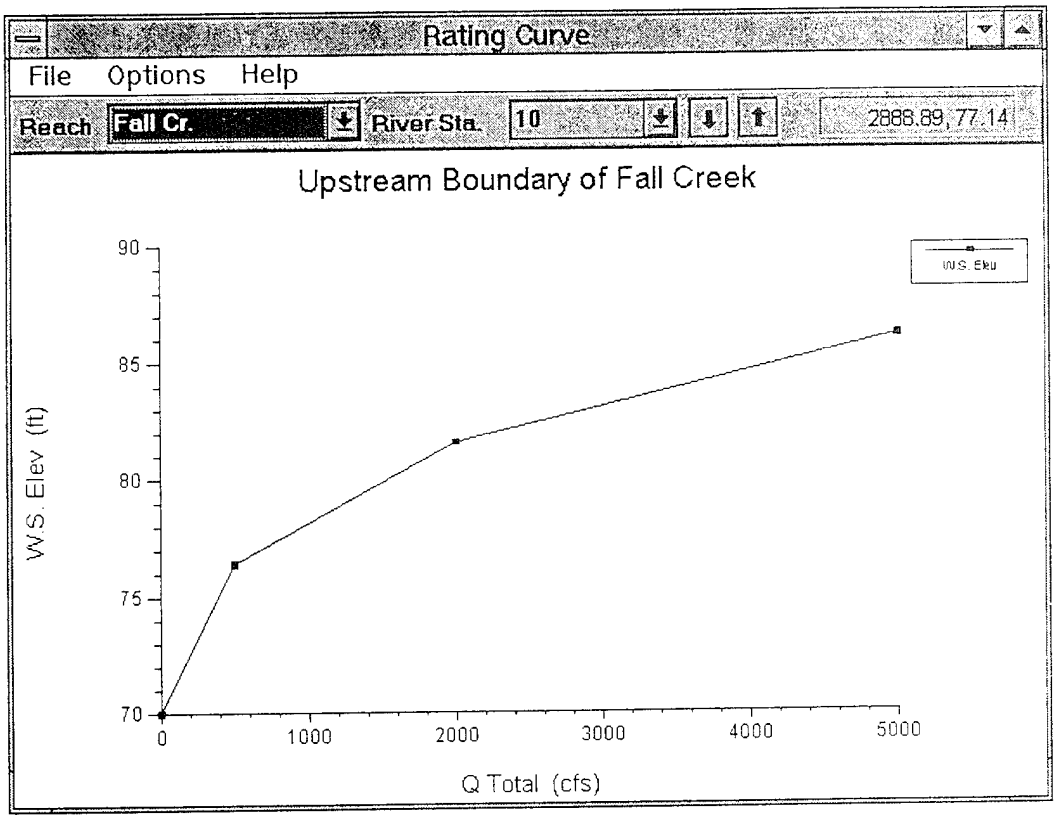


Figure 4.13 Computed Rating Curve for Example Application

Next look at an X-Y-Z Perspective Plot of the river reaches. From the **View** menu bar on the HEC-RAS main window, select **X-Y-Z Perspective Plots.** A multiple cross section perspective plot should appear for the Fall Creek reach as shown in Figure 4.14. Try rotating the perspective view in different directions, and select different reaches to look at.

Now let's look at some tabular output. Go to the **View** menu bar on the HEC-RAS main window. There are two types of tables available, a cross section specific table and a profile table. Select **Cross Section Table** to get the first table to appear. The table should look like the one shown in Figure 4.15. This table shows detailed hydraulic information at a single cross section. Other cross sections can be viewed by selecting the appropriate reach and river mile from the table.

Now bring up the profile table. This table shows a limited number of hydraulic variables for several cross sections. There are several types of profile tables listed under the **Tables** menu bar of the profile table window. Some of the tables are designed to provide specific information at hydraulic structures (e.g. bridges and culverts), while others provide generic information at all cross sections. An example of this type of table is shown in Figure 4.16.

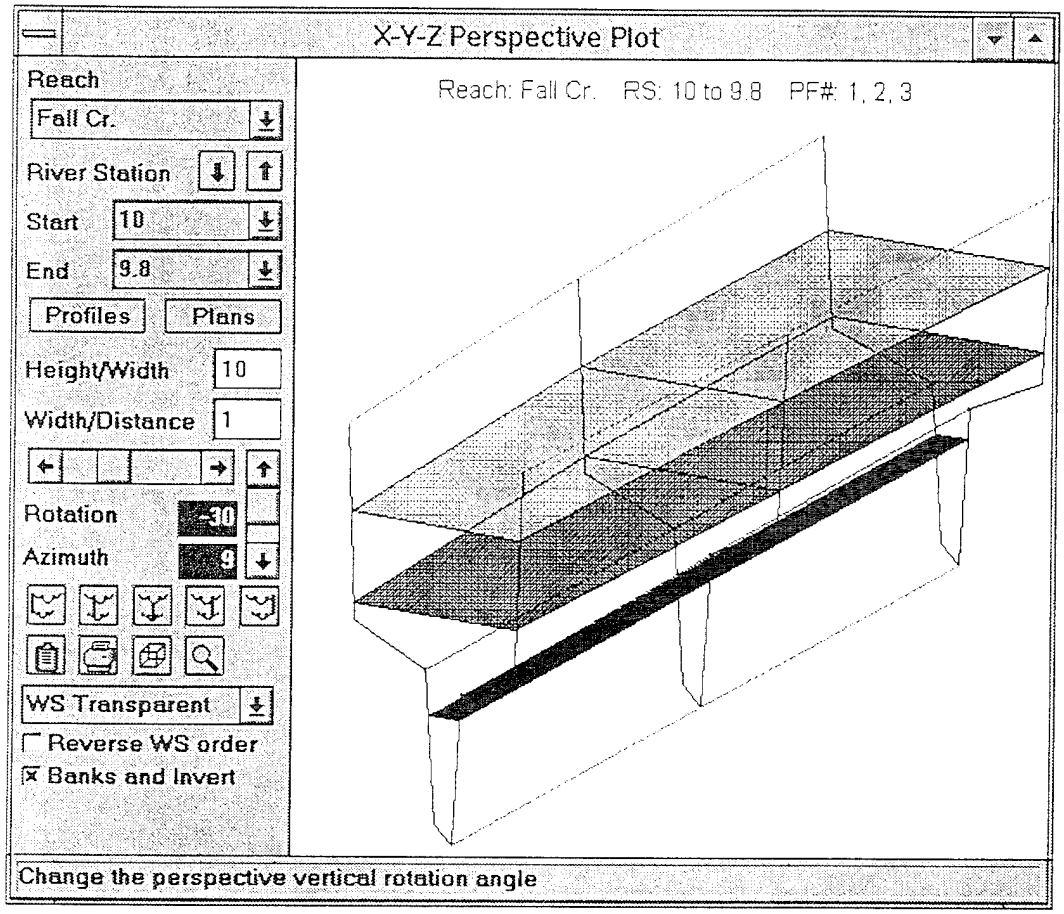


Figure 4.14 X-Y-Z Perspective Plot of Fall Cr. river reach

Cross Section Output

File Type Options

Reach Fall Cr. River Sta. 10 Profile 2

HEC-RAS Plan: Existing Reach: Fall Cr. Riv Sta: 10 Profile: 2					
W.S. Elev (ft)	81.61	Element	Left O.B.	Channel	Right O.B.
Vel Head (ft)	0.23	Wt. n-Val.	0.060	0.035	0.050
E.G. Elev (ft)	81.84	Reach Len. (ft)	450.00	500.00	550.00
E.G. Slope (ft/ft)	0.000647	Flow Area (sq ft)	209.85	364.28	176.96
Q Total (cfs)	2000.00	Flow (cfs)	246.66	1570.71	182.63
Top Width (ft)	232.21	Top Width (ft)	81.61	40.00	110.61
Vel Total (ft/s)	2.66	Avg. Vel. (ft/s)	1.18	4.31	1.03
Max Chl Dpth (ft)	11.61	Hydr. Depth (ft)	2.57	9.11	1.60
Crit W.S. (ft)		Wetted Per. (ft)	82.30	45.64	110.88
Conv. Total (cfs)	78651.9	Conv. (cfs)	9700.2	61769.6	7182.1

Errors, Warnings and Notes

Figure 4.15 Detailed Tabular Output at a Cross Section

Profile Output Table - Standard Table 1

File Options Std. Tables User Tables Help

HEC-RAS Plan: Existing Reach: Fall Cr.

River Sta.	Q Total	Min Ch El	W.S. Elev	Crit W.S.	E.G. Elev	E.G. Slope	Vel Chnl	Flow Area
	(cfs)	(ft)	(ft)	(ft)	(ft)	(ft/ft)	(ft/s)	(sq ft)
10	500.00	70.00	76.44		76.59	0.000771	3.06	163.35
10	2000.00	70.00	81.61		81.84	0.000647	4.31	751.09
10	5000.00	70.00	86.15		86.35	0.000435	4.63	1826.70
9.9	500.00	69.50	76.08		76.22	0.000712	2.97	168.09
9.9	2000.00	69.50	81.30		81.52	0.000599	4.21	754.45
9.9	5000.00	69.50	85.92		86.13	0.000438	4.71	1757.68
9.8	500.00	69.10	75.73		75.87	0.000690	2.94	169.91
9.8	2000.00	69.10	80.97		81.22	0.000631	4.34	690.33
9.8	5000.00	69.10	85.61		85.88	0.000522	5.16	1534.79

Total flow in cross section.

Figure 4.16 Tabular Output in Profile Format

Printing Graphics and Tables

All of the plots and tables can be sent directly to a printer/plotter or passed through the Windows clipboard to another program (e.g. a word processor). The printer or plotter that gets used is based on what you currently have selected as the default printer for Windows. The user has the ability to change many of the default printer settings (e.g. portrait to landscape) before printing occurs.

Sending Graphics Directly to the Printer

To send a graphic to the printer/plotter, do the following:

1. Display the graphic of interest (cross section, profile, rating curve, or river system schematic) on the screen.

2. Using the available options (scaling, labels, grid, etc..), modify the plot to be what you would like printed out.

3. Select **Print Current** from the **File** menu of the displayed graphic. Once Print is selected, a **Printer Options** window will appear, giving the user the opportunity to change any of the default printer settings. Once you have the print settings the way you want them, press the **Print** button on the **Printer Options** window and the plot will automatically be sent to the Windows Print Manager. From that point the Windows Print Manager will control the printing.

Sending Graphics to the Windows Clipboard

To pass a graphic to the Windows clipboard and then to another program, do the following:

1. Display the graphic of interest on the screen.

2. Using the available options, modify the plot to be what you want printed.

3. Select **Copy to Clipboard** from the **File** menu of the displayed graphic. The plot will automatically be sent to the Windows clipboard.

4. Bring up the program that you want to pass the graphic into (e.g. word processor). Select **Paste** from the **Edit** menu of the receiving program. Once the graphic is pasted in, it can be re-sized to the desired dimensions.

Sending Tables Directly to the Printer

To send a table to the printer do the following:

1. Bring up the desired table from the tabular output section of the program.

2. Select **Print** from the **File** menu of the displayed table. Once the Print option is selected, a **Printer Options** window will appear. Set any print options that are desired, then press the **Print** button. This will send the entire table to the Windows Print Manager. From this point the Windows Print Manager will control the printing of the table.

The profile type of tables allow you to print a specific portion of the table, rather than the whole thing. If you desire to only print a portion of the table, do the following:

1. Display the desired profile type table on the screen.

2. Using the mouse, press down on the left mouse button and highlight the area of the table that you would like to print. To get an entire row or column, press down on the left mouse button while moving the pointer across the desired row or column headings.

3. Select **Print** from the **File** menu of the displayed table. Only the highlighted portion of the table and the row and column headings will be sent to the Windows Print Manager.

Sending Tables to the Windows Clipboard

To pass a table to the Windows clipboard and then to another program, do the following:

1. Display the desired table on the screen.

2. Select **Copy to Clipboard** from the **File** menu of the displayed table.

3. Bring up the program that you want to pass the table into. Select **Paste** from the **Edit** menu of the receiving program.

Portions of the profile tables can be sent to the clipboard in the same manner as sending them to the printer.

Practice sending graphics and tables to the printer and the clipboard with the example data set that you currently have open.

Exiting The Program

Before you exit the HEC-RAS software, make sure you have saved all the data. This can be accomplished easily by selecting **Save Project** from the **File** menu on the HEC-RAS main window. Any data (geometric, flow, and plan data) that have not been saved will automatically be saved for you.

To exit the HEC-RAS software, select **Exit** from the **File** menu of the HEC-RAS main window. The program will prompt you to save the project if the data have not been saved previously.

CHAPTER 5

Working With Projects

To create a river hydraulics application with HEC-RAS, you work with projects. A **project** is a collection of files that are used to build a model. This chapter describes projects and how you build and manage them.

Contents

- Understanding Projects
- Elements of a Project
- Creating, Opening, Saving, Renaming, and Deleting Projects
- Project Options

Understanding Projects

As you develop an application, the management of all the files that get created is accomplished through the user interface. When a new project is started, the user is requested to enter a title and filename for the project. All other data are automatically stored by the user interface using the same name as the project file, except for the three character extension. A project consists of:

- One **Project** file (.PRJ)
- One file for each **Plan** (.P01 to .P99)
- One **Run** file for each plan (.R01 to .R99)
- One **Output** file for each plan (.O01 to .O99)
- One file for each set of **Geometry** data (.G01 to .G99)
- One file for each set of **Steady Flow** data (.F01 to .F99)
- One file for each set of **Unsteady Flow** data (.U01 to .U99)
- One file for each set of **Sediment** data (.S01 to .S99)
- One file for each set of **Hydraulic Design** data (.H01 to .H99)

The **Project File** contains: the title of the project; a list of all the files that are associated with the project; and a list of default variables that can be set from the interface. Also included in the project file is a reference to the last plan that the user was working with. This information is updated every time you save the project.

Elements of a Project

The following sections describe the various types of files that can be included in a project. All of these files are either created by the user interface or the various computation engines. The modeler interacts with the data through the user interface, and is not required to create any of these files.

Plan Files

Plan files have the extension .P01 to .P99. The "P" indicates a Plan file, while the number represents the plan number. As plans are created, they are numbered from 01 to 99. The plan file contains: a description and short identifier for the plan; a list of files that are associated with the plan (e.g., geometry file and steady flow file); and a description of all the simulation options that were set for the plan. The plan file is created automatically by the interface each time the user selects **New Plan** or **Save Plan As** from the simulation windows.

Run Files

Run files have the extension .R01 to .R99. The "R" indicates a Run file, while the number represents an association to a particular plan file. A file with an extension of .R01 is the run file that corresponds to the plan file with the extension .P01. The run file contains all of the necessary data to perform the computations that are requested by the associated plan file. For example, if a steady flow analysis is requested, the run file will contain geometry data, steady flow data, and all the necessary computational options that are associated with the plan file. The run file contains the input to any of the computational engines available in the HEC-RAS system. The run file is automatically generated by the interface whenever the user presses the **Compute** button on the Simulation windows. The run file is in an ASCII format, but it is not self explanatory.

Output Files

Output files have the extension .O01 to .O99. The "O" indicates an Output file, while the number represents an association to a particular plan file. A file with the extension .O12 is the output file that corresponds to the plan file with an extension .P12. The output file contains all of the computed results from the requested computational engine. For example, if a steady flow analysis is requested, the output file will contain results from the steady flow computational engine. The output files are in a binary file format and can only be read from the user interface.

Geometry Files

Geometry files have the extension .G01 to .G99. The "G" indicates a Geometry file, while the number corresponds to the order in which they were saved for that particular project. Geometry files contain all of the geometric data for the river system being analyzed. The geometric data consist of: cross section information; hydraulic structures data (e.g. bridges and culverts); coefficients; and modeling approach information. The geometry data are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the-most-part self explanatory. A geometry file is created by the user interface whenever the modeler selects **New Geometry Data** or **Save Geometry Data As** from the Geometric Data window.

Steady Flow Data Files

Steady flow data files have the extension .F01 to .F99. The "F" represents that it is a steady Flow data file, while the number corresponds to the order in which they were saved for that particular project. Steady flow data files contain: the number of profiles to be computed; flow data; and boundary conditions for each reach. The steady flow data files are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the-most-part self explanatory. Steady flow data files are automatically created by the user interface when the modeler selects **New Flow Data** or **Save Flow Data As** from the Steady Flow Data window.

Unsteady Flow Data Files

Unsteady flow data files have the extension .U01 to .U99. The "U" represents that it is an Unsteady flow data file, while the number corresponds to the order in which they were saved for that particular project. Unsteady flow data files contain: flow hydrographs at the upstream boundaries; starting flow conditions; and downstream boundary conditions. The unsteady flow data files are stored in an ASCII format. The file contains key words to describe each piece of data,

and is for-the-most-part self explanatory. Unsteady flow data files are automatically created by the user interface when the modeler selects **New Flow Data** or **Save Flow Data As** from the Unsteady Flow Data window.

Sediment Data Files

Sediment data files have the extension .S01 to .S99. The "S" represents that it is a Sediment data file, while the number corresponds to the order in which they were saved for that particular project. Sediment data files contain: flow data; boundary conditions for each reach; and sediment data. The sediment data files are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the- most-part self explanatory. Sediment data files are automatically created by the user interface when the modeler selects **New Sediment Data** or **Save Sediment Data As** from the Sediment Data window.

Hydraulic Design Data Files

Hydraulic design data files have the extension .H01 to .H99. The "H" represents that it is a Hydraulic design data file, while the number corresponds to the order in which they were saved for that particular project. Hydraulic design data files contain information corresponding to the type of hydraulic design calculation that is requested. The Hydraulic design data files are stored in an ASCII format. The file contains key words to describe each piece of data, and is for-the most-part self explanatory. Hydraulic Design data files are automatically created by the user interface when the modeler selects **New Hydraulic Design Data** or **Save Hydraulic Design Data As** from the Hydraulic Design Data window.

Creating, Opening, Saving, Renaming, and Deleting Projects

The following commands from the **File** menu of the HEC-RAS main window allow you to create, open, save, rename, and delete projects.

File menu command	Description
New Project	Closes the current project, prompting you to save the data if anything has been changed. The user is then prompted to enter a title and filename for the new project.
Open Project	Closes the current project, prompting you to save the data if anything has been changed. Opens an existing project and all of the associated files.
Save Project	Updates the project file and all other files in which data have been modified.
Save Project As	Updates the project file and all other associated data, saving all the information to a filename that you specify.
Rename Project	Allows the user to rename the title of the currently opened project.
Delete Project	Deletes the project file and all other files associated with the selected project. The user is prompted to make sure that they really want to delete all of the files associated with the project.

These commands are the same for all of the other data types that get created by the user interface (Plan data, geometry data, steady flow data, unsteady flow data, sediment data, and hydraulic design data).

Project Options

From the **Options** menu of the main HEC-RAS window, the user can set several default project options. These options include: setting default margins and color control for printing; setting default hydraulic variables; establishing the default units system (English or Metric); and converting existing projects

to a different units system (English to Metric or Metric to English). The following four options are available from the **Options** menu of the main window:

Options menu command	Description
Program Setup	
- BW to Printer	When this option is set all graphics are sent to the printer\plotter in Black and White. When this option is turned off, all graphics are sent as color drawings. Color drawings that are sent to a black and white printer will come out in grey scale shadings.
- BW to Clipboard	When this option is set all graphics are sent to the Windows Clipboard in a Black and White mode. When this option is turned off, the graphics are sent to the Clipboard as color drawings.
- Default Margins	This option allows the user to change the default margins for printing graphics and tables. The default settings are 1 inch margins on all four sides.
- Default File Viewer	This option allows the user to change which program is used for viewing the logfile output. The default is the Windows Write program.
- Open last project	When this option is selected, the program will automatically open the last project worked on, during startup.
Default Parameters	This option allows the user to set defaults for some of the hydraulic variables.
Unit System	This option allows the user to set the default units system to either English or Metric. Once the units system is set, the program assumes that all input data are entered in that units system. Likewise, the display of all output data will be done in the default units system.
Convert Project Units	This option allows the user to convert an existing project from one units system to another. Projects can be converted from English to Metric or from Metric to English.

CHAPTER 6

Entering and Editing Geometric Data

Geometric data consist of establishing the connectivity of the river system (River System Schematic), entering cross-section data, defining all the necessary junction information, adding hydraulic structure data (bridges, culverts, weirs, etc...) and cross section interpolation. The geometric data is entered by selecting **Geometric Data** from the **Edit** menu on the HEC-RAS main window. Once this option is selected, the Geometric Data window will appear as shown in Figure 6.1. The drawing area will be blank on your screen, until you have drawn in your own river system schematic.

This chapter describes how to enter and edit all of the necessary geometric data for a river system.

Contents

- Developing the River System Schematic
- Cross Section Data
- Stream Junctions
- Bridges and Culverts
- Cross Section Interpolation
- Saving the Geometric Data

Developing the River System Schematic

Building The Schematic

The modeler develops the geometric data by first drawing in the river system schematic on the Geometric Data window (Figure 6.1). This is accomplished, on a reach-by-reach basis, by pressing the **River Reach** button and then drawing in a reach from upstream to downstream (in the positive flow direction). After the reach is drawn, the user is prompted to enter an identifier. The identifier can be up to twelve characters in length. As reaches are connected together, junctions are automatically formed by the interface. The modeler is also prompted to enter an identifier for each junction.

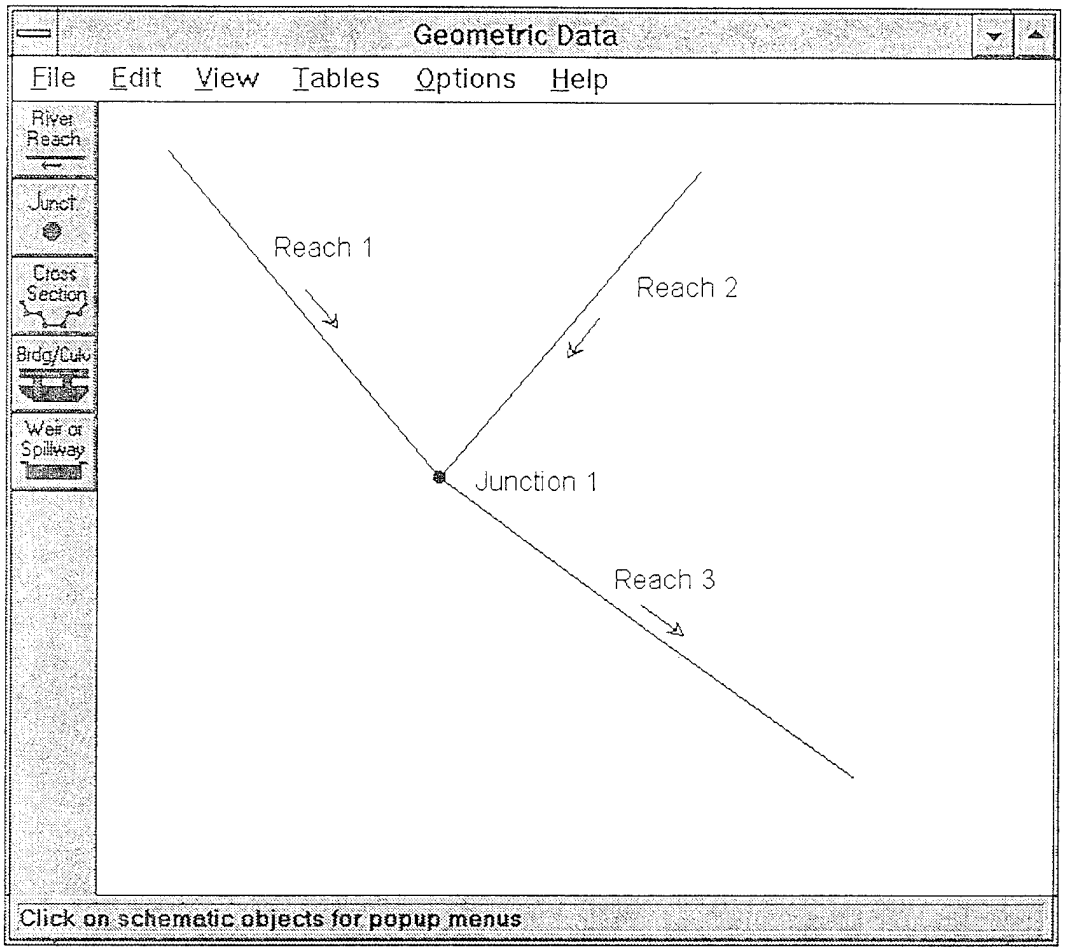


Figure 6.1 Geometric Data Window

Editing The Schematic

There are several options available for editing the river system schematic. These options include: deleting reaches, changing labels, and moving objects (objects are labels, junctions, and ends of reaches). Editing features for the schematic are found under the **Edit** menu of the geometric data window as follows:

Change Name: This option allows the user to change the name of any reach or junction. This is accomplished by first selecting the **Change Name** option from the **Edit** menu, then selecting the particular label that you would like to change. Once you have clicked the left mouse button over the label to be changed, a popup window will appear allowing you to enter a new label. The user can continue to change names by simply selecting the next label to be changed. The **Change Name** option is automatically turned off when the user selects any other option.

Move Object: This option allows you to move any label, junction, or the end of a reach. This is accomplished by first selecting **Move Object** from the **Edit** menu, then selecting the particular object that you would like to move. To select an object and then move it, simply place the mouse pointer over the object, then press the left mouse button down. Move the object to the desired location and then release the left mouse button. The **Move Object** option will remain in effect until the user either turns it off or selects any other option.

Delete Reach: This option is used to delete a reach. This is accomplished by selecting the **Delete Reach** option from the **Edit** menu. A list box containing all the available reaches will appear allowing you to select those reaches that you would like to delete. Be careful when you delete reaches. When you delete a reach, all of its associated data will be deleted also.

Interacting With The Schematic

In addition to modifying the river schematic, there are options available to zoom in, zoom out, and display the cross section river stationing on the schematic. These options are available from the **View** menu on the geometric data window as follows:

Zoom In: This option allows the user to zoom in on a piece of the schematic. This is accomplished by selecting **Zoom In** from the **View** menu, then specifying the area to zoom in on with the mouse. Defining the zoom area is accomplished by placing the mouse pointer in the upper left corner of the desired area. Then press down on the left mouse button and drag the mouse to define a box containing the desired zoom area. Finally, release the left mouse button and the viewing area will display the zoomed in schematic. Also displayed will be a small box in the upper right corner of the viewing area. This box will contain a picture of the entire schematic, with a rectangle defining the area that is zoomed in. In addition to showing you where you are at on the schematic, this zoom box allows you to move around the schematic without zooming out and then back in. To move the zoomed viewing area, simply hold down the left mouse button over the rectangle in the zoom box and move it around the schematic. The zoom box can also be resized. Resizing the zoom box is just like resizing a window.

Zoom Out: This option re-displays the schematic back into its original size before you zoomed in. Zooming out is accomplished by selecting **Zoom Out** from the **View** menu on the geometric data window.

Display River Stationing: This option allows you to display river station tags on the schematic at the locations of any cross section data that have been entered. This is accomplished by selecting **Display River Stationing** from the **View** menu on the geometric data window.

Cross Section Data

After the river system schematic is completed, the next step for the modeler is to enter the cross section data. Cross section data represent the geometric boundary of the stream. Cross sections are located at relatively short intervals along the stream to characterize the flow carrying capacity of the stream and its adjacent floodplain. Cross sections are required at representative locations throughout the stream and at locations where changes occur in discharge, slope, shape, roughness, at locations where levees begin and end, and at hydraulic structures (bridges, culverts, and weirs).

Entering Cross Section Data

To enter cross section data, the user presses the **Cross Section** button on the Geometric Data window (Figure 6.1). Once the cross section button is pressed, the Cross Section Data Editor will appear as shown in Figure 6.2 (except yours will be blank). To add a cross section to the model, the user must do the following:

1. Select the reach that you would like to place the cross section in. This is accomplished by pressing the down arrow on the reach box, and then selecting the reach of choice.

2. Go to the **Options** menu and select **Add a new Cross Section** from the list. An input box will appear prompting you to enter a river station identifier for the new cross section.

3. Enter all of the required data for the new cross section.

4. Enter any desired optional information. Optional cross section information is found under the **Options** menu at the top of the window.

5. Press the **Apply Data** button in order for the interface to accept the information. The apply data button does not save the data to the hard disk, it is only used as a mechanism for telling the interface to use the information that was just entered. If you want the data to be saved to the hard disk you must do that from the **File** menu on the geometric data window.

The required information for a cross section consists of: the reach and river station where the cross section is located; a description; X & Y coordinates (station and elevation points); downstream reach lengths; roughness coefficients; main channel bank stations; and contraction and expansion coefficients. All of the required information is displayed openly on the Cross Section Data editor (Figure 6.2). A description of this information follows:

Reach and River Station. The Reach box allows the user to select a reach from the available reaches in the schematic diagram. The reach label defines which reach the cross section will be located in. The River Station tag defines where the cross section will be located within the specified reach. The river station tag does not have to be the actual river station of the cross section, but it must be a numeric value. Cross sections are ordered in the reach from highest river station upstream to lowest river station downstream. The up and down arrow buttons next to the river station box can be used to sequentially move through the river stations.

Description. The description box is used to describe the cross section location in more detail than just the reach and river station. This box has a limit of 256 characters. Only the first line of information is displayed, unless the button to the right of the box is pressed. Also, the first 40 characters of the description is used as a label for cross section plots and cross section tables.

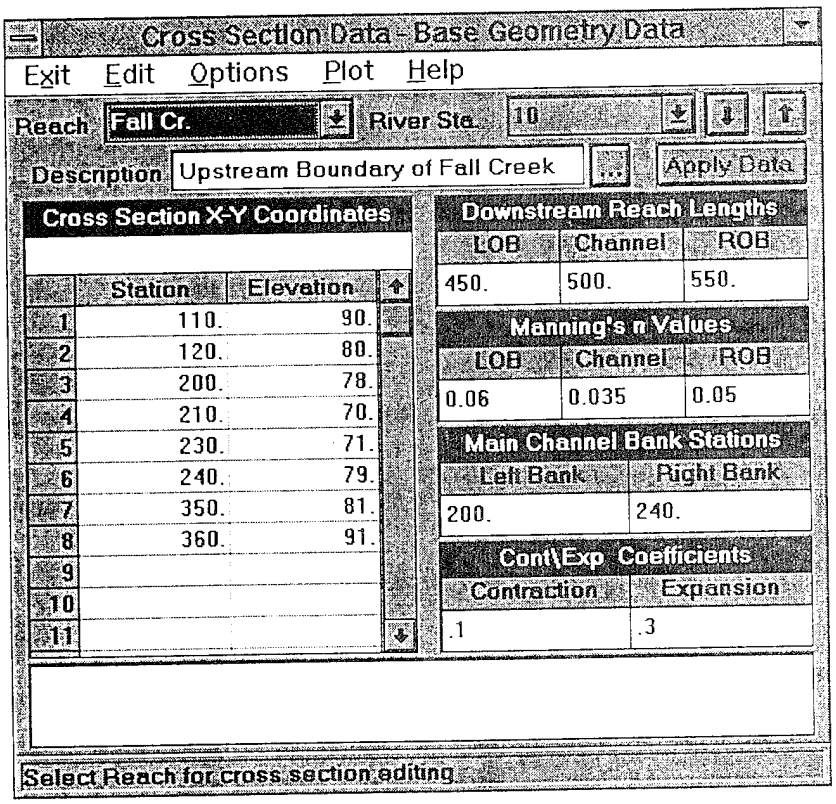


Figure 6.2 Cross Section Data Editor

Cross Section X & Y Coordinates. This table is used to enter the station and elevation information of the cross section. Station and elevation information is entered in feet (meters for metric).

Downstream Reach Lengths. The downstream reach lengths describe the distance between the current cross section and the next cross section downstream. Reach lengths are defined for the left overbank, main channel, and the right overbank. Reach lengths are entered in feet (meters for metric).

Manning's n Values. At a minimum the user must specify manning's n values for the left overbank, main channel, and the right overbank. Alternative roughness coefficient options are available from the **Options** menu.

Main Channel Bank Stations. The main channel bank stations are used to define what portion of the cross section is considered the main channel and what is considered left and right overbank area. The bank stations must correspond to stations entered on the cross section X & Y coordinates table. If the user enters a value that does not correspond to the station points of the cross section, the interface will ask the user if they would like the value to be automatically interpolated and added to the cross section data.

Contraction & Expansion Coefficients. Contraction and expansion coefficients are use to evaluate the amount of energy loss that occurs because of a flow contraction or expansion. The coefficients are multiplied by the change in velocity head from the current cross section and the previous cross section.

Once all of the required data for the cross section are entered, make sure you press the **Apply data** button to ensure that the interface accepts the data that was just entered.

Editing Cross Section Data

The bulk of the cross section data is the station and elevation information. There are several features available under the **Edit** menu to assist the user in modifying this information. These features include the following:

Undo Editing. This editing feature applies to all of the information on the cross section data editor. Once data has been entered and the **Apply Data** button has been pressed, the **Undo Editing** feature is activated. If any changes are made from this point, the user can get the original information back by selecting the **Undo Edit** option from the **Edit** menu. Once the **Apply Data** button is pressed, the new information is considered good and the **Undo Edit** feature is reset to the new data.

Cut, Copy, and Paste. Cut, Copy, and Paste features are available for the station and elevation information on the cross section editor. These features allow the user to pass cross section station and elevation data to and from the Windows Clipboard. To use this feature, first highlight a cell or multiple cells on the station and elevation table. Cells are highlighted by pressing down on the left mouse button and moving it over the cells that you would like to be highlighted. Next select either the **Cut** or **Copy** feature from the **Edit** menu. If **Cut** is selected, the information is placed in the Windows Clipboard and then it is deleted from the table. If **Copy** is selected, the information is placed in the Windows Clipboard, but it also remains in the table. Once the information is in the Windows Clipboard it can be pasted into the station and elevation table of any cross section. To paste data into another cross section, first go to the cross section in which you would like the data to be placed. Highlight the area of the table in which you want the data to be placed. Then select the **Paste** option from the **Edit** menu. The cut, copy, and paste features can also be used to pass station and elevation information between HEC-RAS and other programs.

Delete. This option allows the user to delete a single cell or multiple cells in the station/elevation table. Once the cells are deleted, everything below those cells is automatically moved up. To use this option, first highlight the cells that you would like to delete, then select the **Delete** option from the **Edit** menu. If you would like to clear cells, without moving the data below those cells, simply highlight the cells and press the delete key.

Insert. This option allows the user to insert one or several rows in the middle of existing data in the station/elevation table. To use this option, first highlight the area in the table that you would like to be inserted. The select **Insert** from the **Edit** menu. The rows will be inserted and all of the data will be moved down the appropriate number of rows. The user can also insert a single row by placing the curser in the row just below where you would like the new row to be inserted. Then select **Insert** from the **Edit** menu. The row will be inserted and all of the data below the current row will be moved down one row.

Cross Section Options

Information that is not required, but is optional, is available from the **Options** menu at the top of the cross section data editor window (Figure 6.2). Options consist of the following:

Add a new Cross Section. This option initiates the process of adding a cross section to the data set. The user is prompted to enter a river station tag for the new cross section. The river station tag locates the cross section within the selected reach. Once the river station is entered, the cross section data editor is cleared and the user can begin entering the data for the cross section.

Copy Current Cross Section. This option allows the user to make a copy of the cross section that is currently displayed in the editor. When this option is selected, the user is prompted to select a reach and enter a river station for the new section. Once the information is entered, the new cross section is displayed in the editor. At this point it is up to the user to change the description and any other information about the cross section. This option is normally used to make interpolated cross sections between two surveyed cross sections. Once the section is copied, the user can adjust the elevations and stationing of the cross section to adequately depict the geometry between the two surveyed sections.

Rename Cross Section. This option allows the user to change the River Station of the currently displayed cross section.

Delete Cross Section. This option will delete the currently displayed cross section. The user is prompted with a message stating specifically which cross section is going to be deleted, and requesting the user to press the **OK** button or the **Cancel** button.

Adjust Elevations. This option allows the user to adjust all of the elevations of the currently displayed cross section. Positive or negative elevation changes can be entered. Once the value is entered, the interface automatically adjusts all the elevations in the table.

Adjust Stations. This option allows the user to adjust the stationing of the currently displayed cross section. Two options are available. The first option **(Multiply by a Factor)** allows the user to separately expand and/or contract the left overbank, main channel, and the right overbank. When this option is selected, the user is prompted to enter a multiplier for each of the three flow elements (left overbank, main channel, and right overbank). If the multiplier is less than one, the flow element is contracted. If the multiplier is greater than one, the flow element is expanded. Once the information is entered, and the user hits the **OK** button, the interface automatically performs the contraction and/or expansions. The cross section should be reviewed to ensure that the desired adjustments were performed. The second option (**Add a Constant**) allows the user to add or subtract a constant value from all the stations in the cross section. This would allow the entire cross section to be shifted to the right or the left.

Adjust n Values. This option allows the user to either increase or decrease all the n values of the current cross section. The user is prompted for a single value. This value is then used as the multiplier for all of the n values of the current cross section.

Ineffective Flow Areas. This option allows the user to define areas of the cross section that will contain water that is not actively being conveyed (ineffective flow). Ineffective flow areas are often used to describe portions

of a cross section in which water will pond, but the velocity of that water, in the downstream direction, is close to zero. This water is included in the storage calculations and other wetted cross section parameters, but it is not included as part of the active flow area. When using ineffective flow areas, no additional wetted perimeter is added to the active flow area. An example of an ineffective flow area is shown in Figure 6.3. The cross-hatched area on the left of the plot represents the ineffective flow area.

Two alternatives are available for setting ineffective flow areas. The first option allows the user to define a left station and elevation and a right station and elevation (normal ineffective areas). When this option is used, and if the water surface is below the established ineffective elevations, the areas to the left of the left station and to the right of the right station are considered ineffective. Once the water surface goes above either of the established elevations, then that specific area is no longer considered ineffective.

The second option allows for the establishment of blocked ineffective flow areas. Blocked ineffective flow areas require the user to enter an elevation, a left station, and a right station for each ineffective block. Up to five blocked ineffective flow areas can be entered at each cross section. Once the water surface goes above the elevation of the blocked ineffective flow area, the blocked area is no longer considered ineffective.

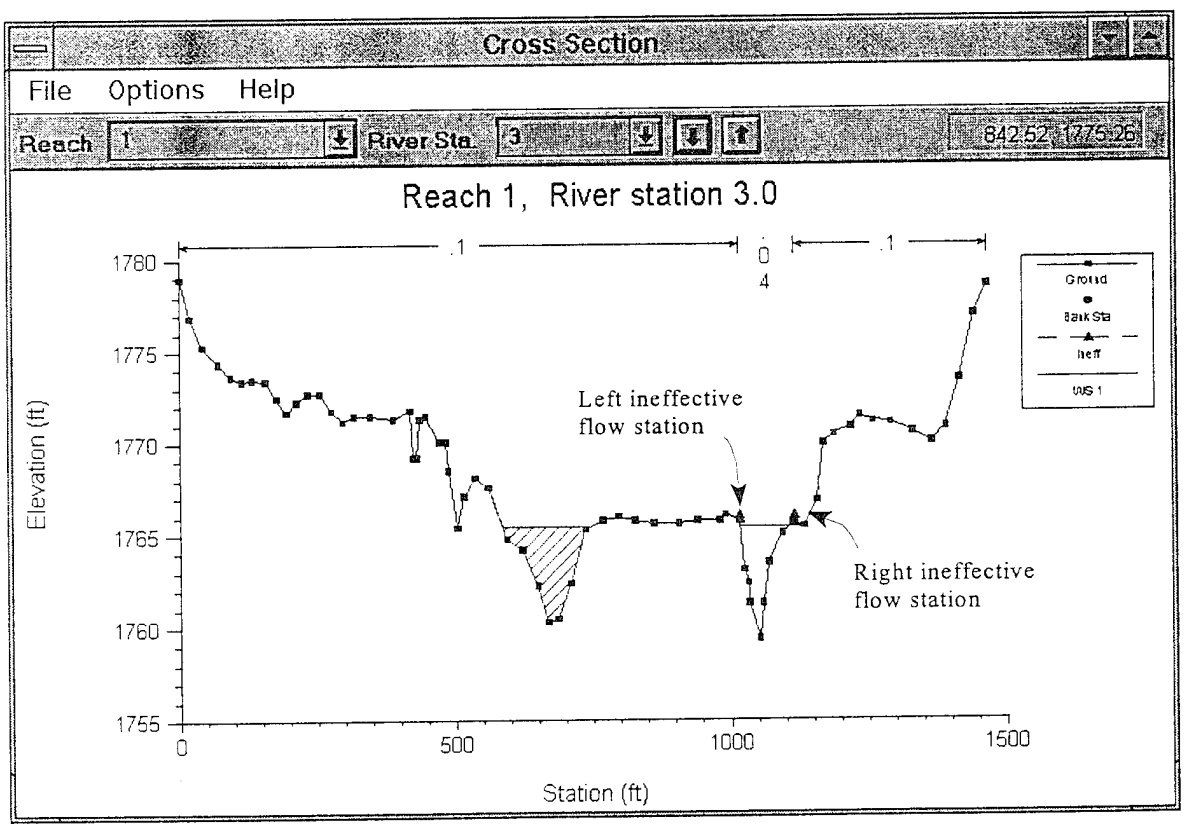


Figure 6.3 Cross section with ineffective flow areas

Levees. This option allows the user to establish a left and/or right levee station and elevation on any cross section. When levees are established, no water can go to the left of the left levee station or to the right of the right levee station until either of the levee elevations are exceeded. Levee stations must be defined explicitly, or the program assumes that water can go anywhere within the cross section. An example of a cross section with a levee on the left side is shown in Figure 6.4. In this example the levee station and elevation is associated with an existing point on the cross section.

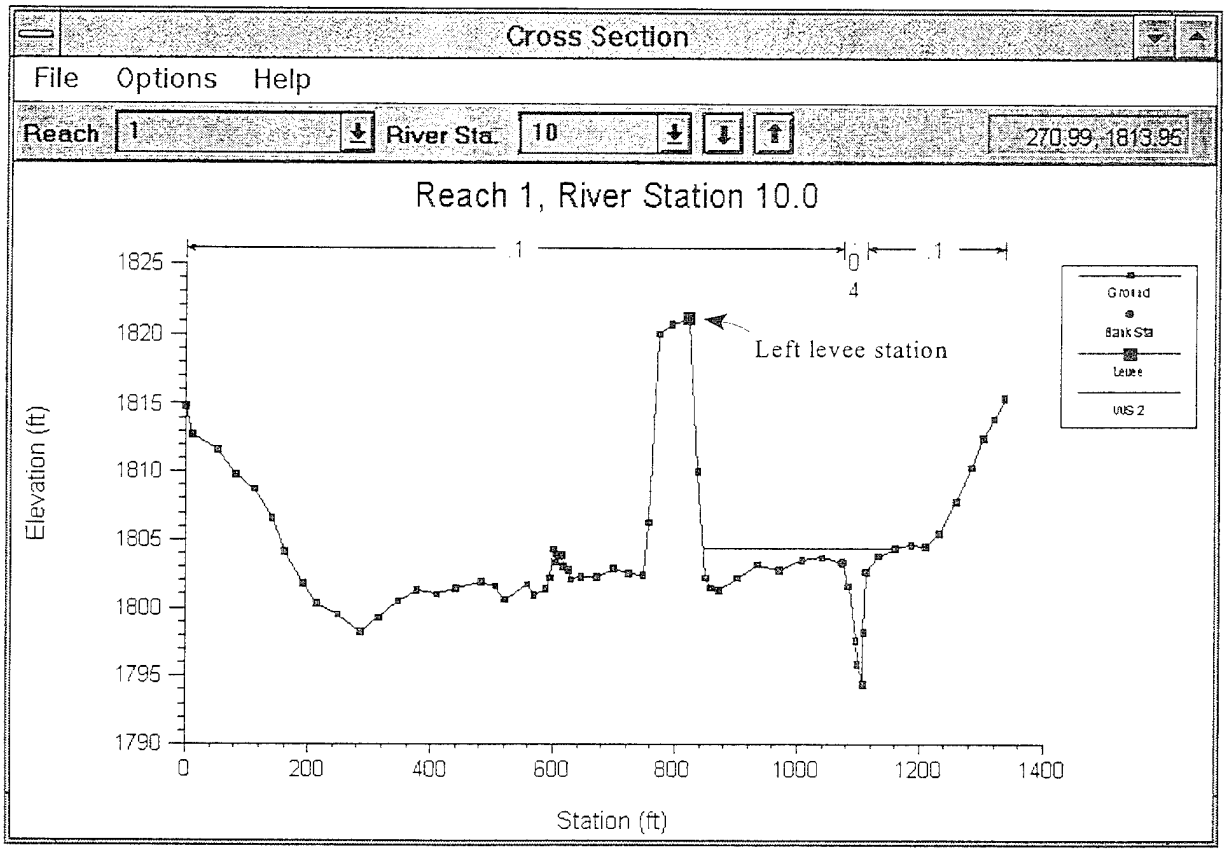


Figure 6.4 Example of the Levee Option

The user may want to add levees into a data set in order to see what effect a levee will have on the water surface. A simple way to do this is to set a levee station and elevation that is above the existing ground. If a levee elevation is placed above the existing geometry of the cross section, then a vertical wall is placed at that station up to the established levee height. Additional wetted perimeter is included when water comes into contact with the levee wall. An example of this is shown in Figure 6.5.

Blocked Obstructions. This option allows the user to define areas of the cross section that will be permanently blocked out. Blocked obstructions decrease flow area and add wetted perimeter when the water comes in contact with the obstruction. A blocked obstruction does not prevent water

from going outside of the obstruction.

Two alternatives are available for entering blocked obstructions. The first option allows the user to define a left station and elevation and a right station and elevation (normal blocked areas). When this option is used, the area to the left of the left station and to the right of the right station will be completely blocked out. An example of this type of blocked obstruction is shown in Figure 6.6.

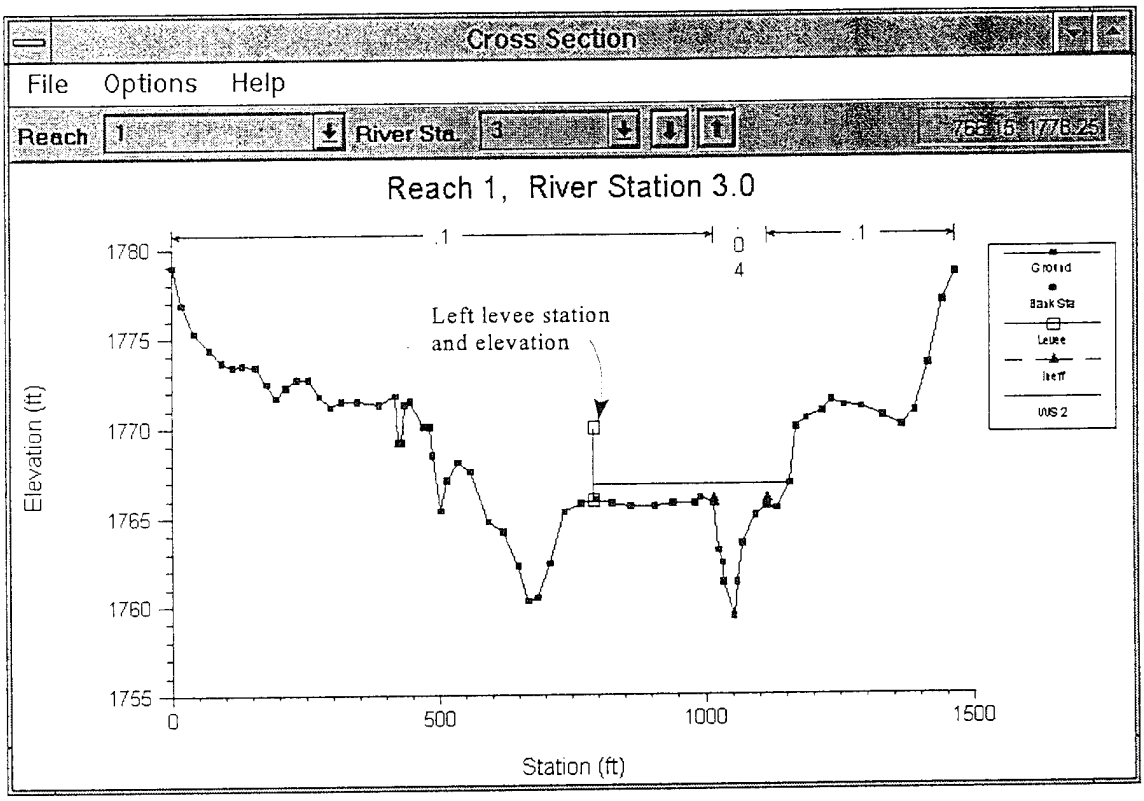


Figure 6.5 Example Levee Added to a Cross Section

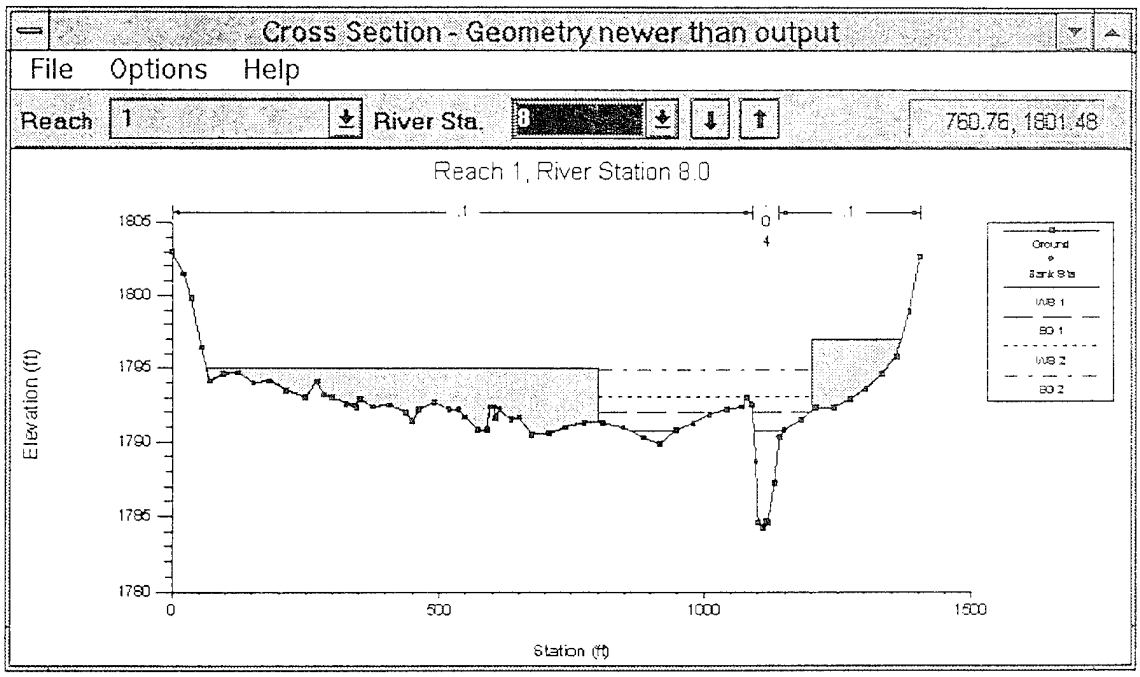


Figure 6.6 Example of Normal Blocked Obstructions

The second option, for blocked obstructions, allows the user to enter up to 20 individual blocks (Multiple Blocks). With this option the user enters a left station, a right station, and an elevation for each of the blocks. An example of a cross section with multiple blocked obstructions is shown in Figure 6.7.

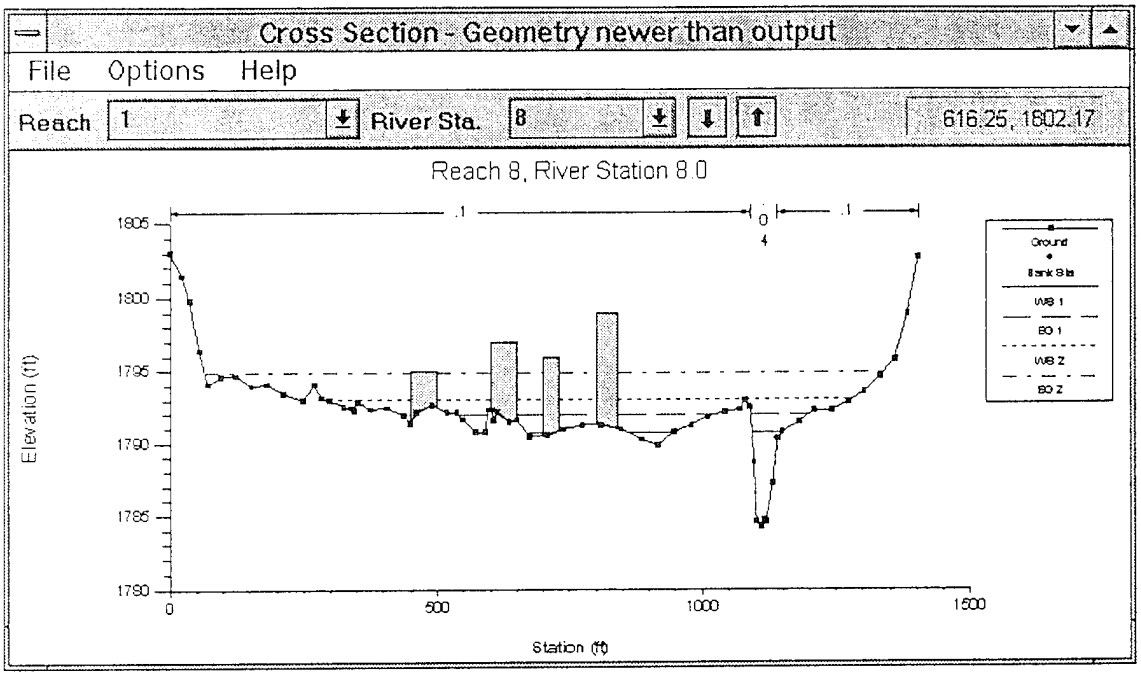


Figure 6.7 Example of a Cross Section With Blocked Obstructions

Add a Lid to XS. This option allows the user to add lid (similar to a bridge deck/roadway) to any cross section. This is commonly used when trying to model a long tunnel. The ground geometry can be used to describe the bottom half of the tunnel, while the lid can describe the top half. A lid can be added to any number of sections in a row. The program treats cross sections with lids just like any other cross section. The energy equation is used to balance a water surface, with the assumption of open channel flow. The only difference is that the program will subtract out area and add wetted perimeter when the water surface comes into contact with the lid.

Horizontal Variation in n Values. This option allows the user to enter more than three Manning's n values for the current cross section. When this option is selected, an additional column for n values is added to the cross section coordinates table as shown in Figure 6.8. A Manning's n value must be placed in the first row of the table. This n value is good for all cross section stations until a new n value shows up in the table. The user does not have to enter an n value for every station, only at the locations where the n value is changing.

Set Maximum Sta./Elev. Points. This option allows the user to set the maximum number of station and elevation points in a cross section. The default value is set to 100 points. A maximum value of 500 can be set. This option is only necessary when the user wants to enter more than 100 points for the station and elevation points of a cross section.

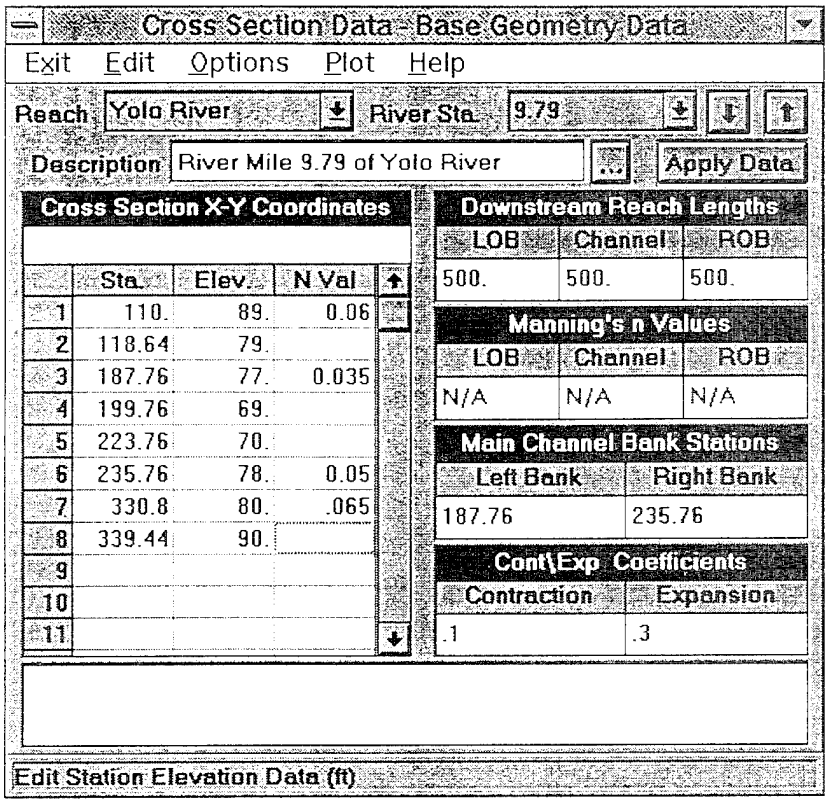


Figure 6.8 Cross section with horizontal variation of n values selected

Plotting Cross Section Data

Once all the data have been entered for a cross section, you should plot the cross section to inspect it for possible data errors. To plot the current cross section from the cross section editor, select **Plot Cross Section** from the **Plot** menu.

Stream Junctions

Entering Junction Data

Stream junctions are defined as locations where two or more streams come together or split apart. Junction data consist of a description, reach lengths across the junction, tributary angles, and modeling approach. To enter junction data the user presses the **Junction** button on the Geometric Data window (Figure 6.1). Once the junction button is pressed, the junction editor will appear as shown in Figure 6.9.

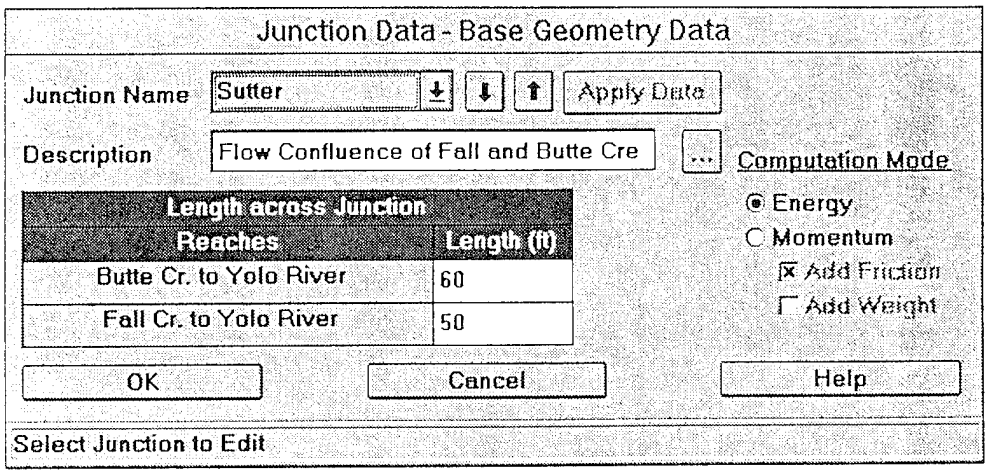


Figure 6.9 Junction Data Editor

The junction editor will come up with one of the junctions loaded. Fill out the description and reach lengths for the junction. Reach lengths across the junction are entered here instead of the cross section data editor. This allows for the lengths across very complicated confluences (i.e. flow splits) to be accommodated. In the cross section data, the reach lengths for the downstream cross section of each reach should be left blank or set to zero.

Selecting A Modeling Approach

In HEC-RAS a junction can be modeled by either the energy equation or the momentum equation. The energy equation does not take into account the angle of a tributary coming in or leaving, while the momentum equation does. In most cases the amount of energy loss due to the angle of the tributary flow is not significant, and using the energy equation to model the junction is more than adequate. However, there are situations where the angle of the tributary can cause significant energy losses. In these situations it would be more appropriate to use the momentum approach. When the momentum approach is selected, an additional column is added to the table next to the junction lengths. This column is used to enter an angle for any river reach that is coming into or exiting the main river. For the reaches that are considered to be the main river, the angle should be left blank or set to zero. Also, the user has the option to turn friction and weight forces on or off during the momentum calculations. The default is to have the weight force turned off.

If there is more than one junction in the river schematic, the other junctions can be selected from the Junction Name box at the upper left corner of the window. Enter all the data for each junction in the river system, then close the window by pressing the **OK** button in the lower left corner of the window. When the junction data editor is closed the data are automatically applied.

Bridges and Culverts

Once all of the necessary cross-section data have been entered, the modeler can then add any bridges or culverts that are required. HEC-RAS computes energy losses caused by structures such as bridges and culverts in three parts. One part consists of losses that occur in the reach immediately downstream from the structure where an expansion of flow takes place. The second part is the losses at the structure itself, which can be modeled with several different methods. The third part consists of losses that occur in the reach immediately upstream of the structure where the flow is contracting to get through the opening.

The bridge routines in HEC-RAS allow the modeler to analyze a bridge with several different methods without changing the bridge geometry. The bridge routines have the ability to model low flow (Class A, B, and C), low flow and weir flow (with adjustments for submergence), pressure flow (orifice and sluice gate equations), pressure and weir flow, and high flows with the energy equation only. The model allows for multiple bridge and/or culvert openings at a single location.

The culvert hydraulics in HEC-RAS are based on the Federal Highway Administrations (FHWA) standard equations from the publication Hydraulic Design of Highway Culverts (FHWA, 1985). The culvert routines include the ability to model circular, box, elliptical, arch, pipe arch, and semi circular culverts. The HEC-RAS program has the ability to model multiple culverts at a single location. The culverts can have different shapes, sizes, elevations, and loss coefficients. The user can also specify the number of identical barrels for each culvert type.

Cross Section Locations

The bridge and culvert routines utilize four user defined cross sections in the computations of energy losses due to the structure. A plan view of the basic cross section layout is shown in Figure 6.10.

Cross section 1 is located sufficiently downstream from the structure so that the flow is not affected by the structure (i.e. the flow has fully expanded). This distance should generally be determined by field investigation during high flows. If field investigation is not possible, then there are two sets of criteria for locating the downstream section. The USGS has established a criterion of locating cross section 1 a distance downstream from the bridge as equal to one times the bridge opening width. Traditionally the Corps of Engineers criterion has been to locate the downstream cross section about four times the average length of the side constriction caused by the structure abutments. In practical applications this expansion distance will vary depending upon the degree of constriction, the shape of the constriction, the magnitude of the flow, and the velocity of the flow. Both of the two criteria

should be used as guidance for placing cross section 1. The user should not allow the distance between cross section 1 and 2 to become so great that friction losses will not be adequately modeled. If the modeler feels that the expansion reach will require a long distance, then intermediate cross sections should be placed within the expansion reach in order to adequately model friction losses.

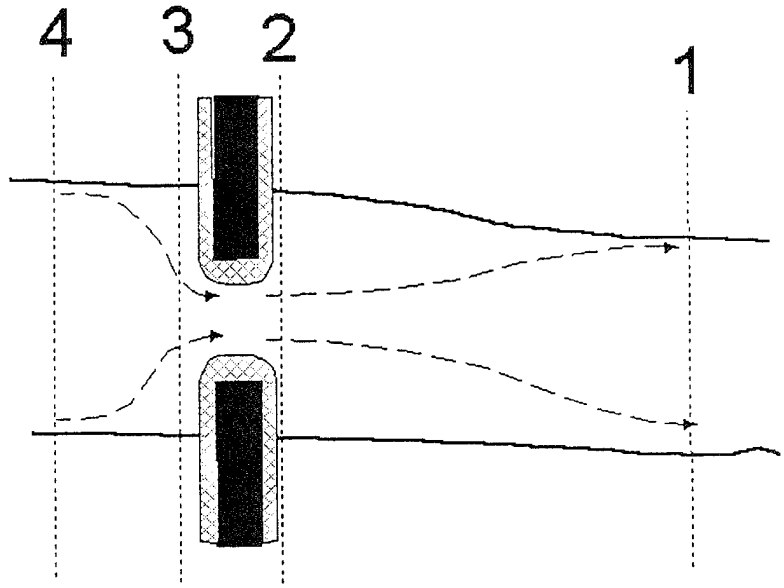


Figure 6.10 Cross Section Locations at a Bridge or Culvert

Cross section 2 is located immediately downstream from the bridge (i.e. within a few feet). This cross section should represent the effective flow just outside the bridge.

Cross section 3 should be located just upstream from the bridge. The distance between cross section 3 and the bridge should be relatively short. This distance should only reflect the length required for the abrupt acceleration and contraction of the flow that occurs in the immediate area of the opening. Cross section 3 represents the effective flow area just upstream of the bridge.

Both cross sections 2 and 3 will have ineffective flow areas to either side of the bridge opening during low flow and pressure flow profiles. In order to model only the effective flow areas at these two sections, the modeler should use the ineffective flow area option. This option is selected from the cross section data editor.

Cross section 4 is an upstream cross section where the flow lines are approximately parallel and the cross section is fully effective. Because flow contractions can occur over a shorter distance than flow expansions, the distance between cross section 3 and 4 should be roughly one times the average width of the opening. However, this criterion for locating the upstream cross section may result in too short a reach length for situations where the width of the bridge opening is very small in comparison to the floodplain. An alternative criterion would be to locate the cross section a distance upstream equal to the average contraction width.

During the hydraulic computations, the program automatically formulates two additional cross sections inside of a bridge structure. This is not necessary for culverts. The geometry inside of the bridge is a combination of the bounding cross sections (2 and 3) and the bridge geometry. The bridge geometry consists of the bridge deck, abutments if necessary, and any piers that may exist. The user can specify different bridge geometry for the upstream and downstream sides of the structure if necessary. Cross section 2 and the structure information on the downstream side are used as the geometry just inside the structure at the downstream end. Cross section 3 and the upstream structure information are used as the bridge geometry just inside the structure at the upstream end.

Contraction and Expansion Losses

Losses due to the contraction and expansion of flow between cross sections are determined by the standard step profile calculations. Manning's equation is used to calculate friction losses, and all other losses are described in terms of coefficient times the absolute value of the change in velocity head between adjacent cross sections. When the velocity head increases in the downstream direction a contraction coefficient is used; and when the velocity head decreases in the downstream direction, an expansion coefficient is used.

Bridge Hydraulic Computations

Low Flow Computations. For low flow computations the program first uses the momentum equation to identify the class of flow. This is accomplished by first calculating the momentum at critical depth inside the bridge at the upstream and downstream ends. The end with the higher momentum (therefore most constricted section) will be the controlling section in the bridge. The momentum at critical depth in the controlling section is then compared to the momentum of the flow downstream of the bridge when performing a subcritical profile (upstream of the bridge for a supercritical profile). If the momentum downstream is greater than the critical depth momentum inside the bridge, the class of flow is considered to be completely subcritical (i.e. class A low flow). If the momentum downstream is less than the momentum at critical depth in the bridge, then it is assumed that the

constriction will cause the flow to pass through critical depth and a hydraulic jump will occur at some distance downstream (i.e. class B low flow). If the profile is completely supercritical through the bridge then this is considered class C low flow. Depending on the class of flow the program will do the following:

Class A low flow. Class A low flow exists when the water surface through the bridge is completely subcritical (i.e. above critical depth). Energy losses through the expansion (sections 2 to 1) are calculated as friction losses and expansion losses. Friction losses are based on a weighted friction slope times a weighted reach length between sections 1 and 2. The average friction slope is based on one of the four available alternatives in HEC-RAS, with the average-conveyance method being the default. This option is user selectable. The average length used in the calculation is based on a discharge-weighted reach length.

There are four methods available for computing losses through the bridge (sections 2 to 3):

- Energy equation (standard step method)
- Momentum balance
- Yarnell equation
- USGS Contracted Opening method

The user can select any or all of these methods in the computations. If more than one method is selected, the user must choose either a single method as the final solution or tell the program to use the method that computes the greatest energy loss through the bridge as the answer at section 3. This allows the modeler to compare the answers from several techniques all in a single execution of the program. Minimal results are available for all the methods computed, but detailed results are available for the method that is selected as the final answer.

Energy losses through the contraction (sections 3 to 4) are calculated as friction losses and contraction losses. Friction and contraction losses between sections 3 and 4 are calculated the same as friction and expansion losses between sections 1 and 2.

Class B low flow. Class B low flow can exist for either subcritical or supercritical profiles. For either profile, class B flow occurs when the profile passes through critical depth in the bridge constriction. For a **subcritical profile**, the momentum equation is used to compute an upstream water surface above critical depth and a downstream water surface below critical depth, using a momentum balance through the bridge. For a **supercritical profile,** the bridge is acting as a control and is causing the upstream water surface

elevation to be above critical depth. Momentum is used again to calculate an upstream water surface above critical depth and a downstream water surface below critical depth. The program will proceed with forewater calculations downstream from the bridge.

Class C low flow. Class C low flow exists when the water surface through the bridge is completely supercritical. The program can use either the energy equation or the momentum equation to compute the water surface through the bridge.

Pressure Flow Computations. Pressure flow occurs when the flow comes into contact with the low chord of the bridge. Once the flow comes into contact with the upstream side of the bridge, a backwater occurs and orifice flow is established. The program will handle two cases of orifice flow: the first is when only the upstream side of the bridge is in contact with the water; and the second is when the bridge constriction is flowing completely full. For the first case, a sluice gate type of equation is used, as described in "Hydraulics of Bridged Waterways" (FHWA, 1978). In the second case, the standard full flowing orifice equation is used. The program will begin checking for the possibility of pressure flow when the energy grade line goes above the maximum low chord elevation. Once pressure flow is computed, the pressure flow answer is compared to the low flow answer and the higher of the two is used. The user has the option to tell the program to use the water surface, instead of energy, to trigger the pressure flow calculation.

Weir Flow Computations. Flow over the bridge and the roadway approaching the bridge will be calculated using the standard weir equation. For high tailwater elevations the program will automatically reduce the amount of weir flow to account for submergence on the weir. This is accomplished by reducing the weir coefficient based on the amount of submergence. When the weir becomes highly submerged, the program will automatically switch to calculating losses based on the energy equation (standard step backwater). The criteria for when the program switches to energy based calculations is user controllable.

Combination Flow. Sometimes combinations of low flow or pressure flow occur with weir flow. In these cases an iterative procedure is used to determine the amount of each type of flow.

Entering and Editing Bridge Data

To enter bridge data the user presses the **Bridge/Culvert** button on the geometric data window (Figure 6.1). Once the bridge/culvert button is pressed, the Bridge/Culvert Data Editor will appear as shown in Figure 6.11. To add a bridge to the model the user must do the following:

1. Select the reach that you would like to place the bridge in. This is accomplished by pressing the down arrow on the reach box, and then selecting the reach of choice.

2. Go to the **Options** menu and select **Add a Bridge and/or Culvert** from the list. An input box will appear prompting you to enter a river station identifier for the new bridge.

3. Enter all of the required data for the new bridge. This may include:
   - Bridge Deck
   - Abutments
   - Piers
   - Bridge modeling approach information

4. Enter any desired optional information. Optional bridge information is found under the **Options** menu at the top of the window.

5. Press the **Apply Data** button for the interface to accept the data.

The required information for a bridge consists of: the reach and river station where the bridge is located; a short description of the bridge; the bridge deck; bridge abutments (if they exist); bridge piers (if the bridge has piers); and specifying the bridge modeling approach. A description of this information follows:

Reach and River Station. The Reach box allows the user to select a reach from the available reaches that are defined in the schematic diagram. The reach label defines which reach the bridge will be located in. The River Station tag defines where the bridge will be located within the specified reach. The river station tag does not have to be the actual river station of the bridge, but it must be a numeric value. The river station tag for the bridge should be numerically between the two cross sections that bound the bridge. Once the user selects **Add a Bridge and/or Culvert** from the options menu, an input box will appear prompting you to enter a river station tag for the new bridge. After the river station tag is entered, the two cross sections that bound the bridge will be displayed on the editor.

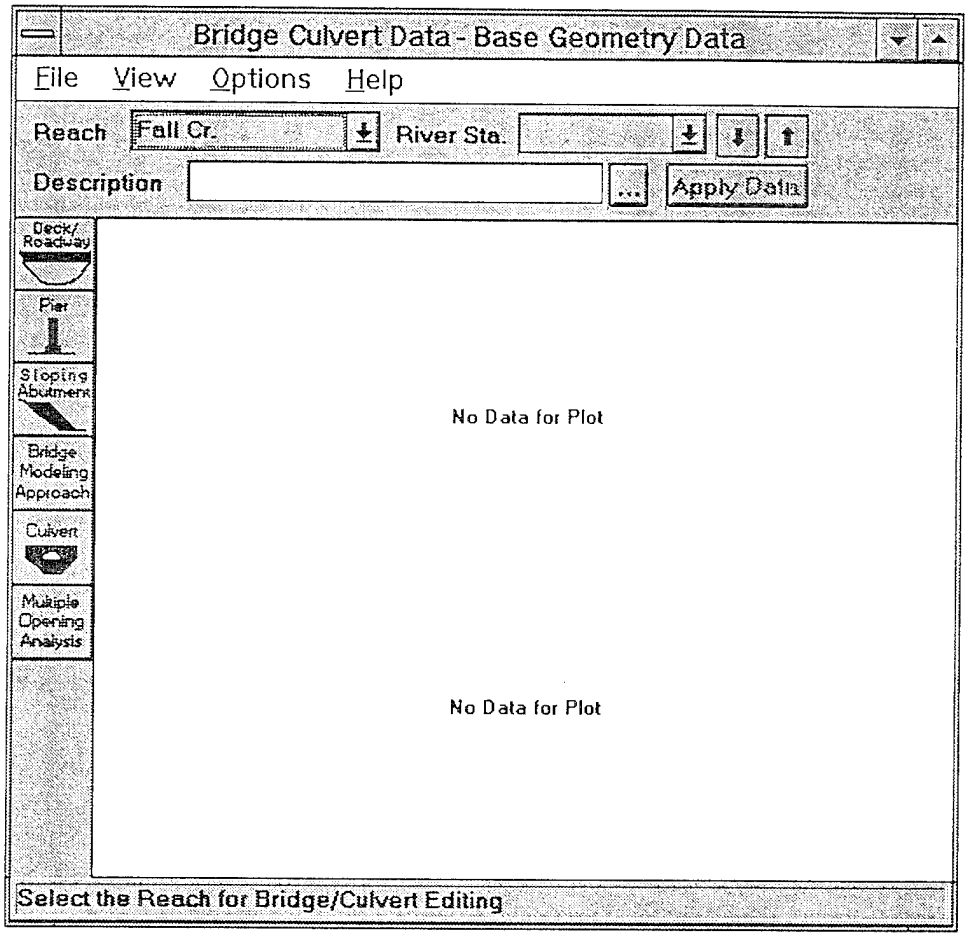


Figure 6.11 Bridge/Culvert Data Editor.

Description. The description box is used to describe the bridge location in more detail than just the reach and river station. This box has a limit of 256 characters. Only the first line of information is displayed, unless the button to the right of the box is pressed. Also, the first 40 characters of the description are used as a label for bridge plots and tables.

Bridge Deck/Roadway. The bridge deck editor is used to describe the area that will be blocked out due to the bridge deck, road embankment and vertical abutments. To enter bridge deck information the user presses the **Deck** button on the Bridge/Culvert Data Editor. Once the deck button is pressed, the Deck Editor will appear as in Figure 6.12 (except yours will be blank). The information entered in the deck editor consists of the following:

Distance - The distance field is used to enter the distance between the upstream side of the bridge deck and the cross section immediately upstream of the bridge.

Width - The width field is used to enter the width of the bridge deck/roadway. The distance between the bridge deck and the downstream bounding cross

section will equal the main channel reach length minus the sum of the bridge "width" and the "distance" between the bridge and the upstream section.

Weir Coefficient - Coefficient that will be used for weir flow over the bridge deck in the standard weir equation.

Skew Angle - Angle that the bridge deck is skewed from a line perpendicular to the flow lines passing through the bridge.

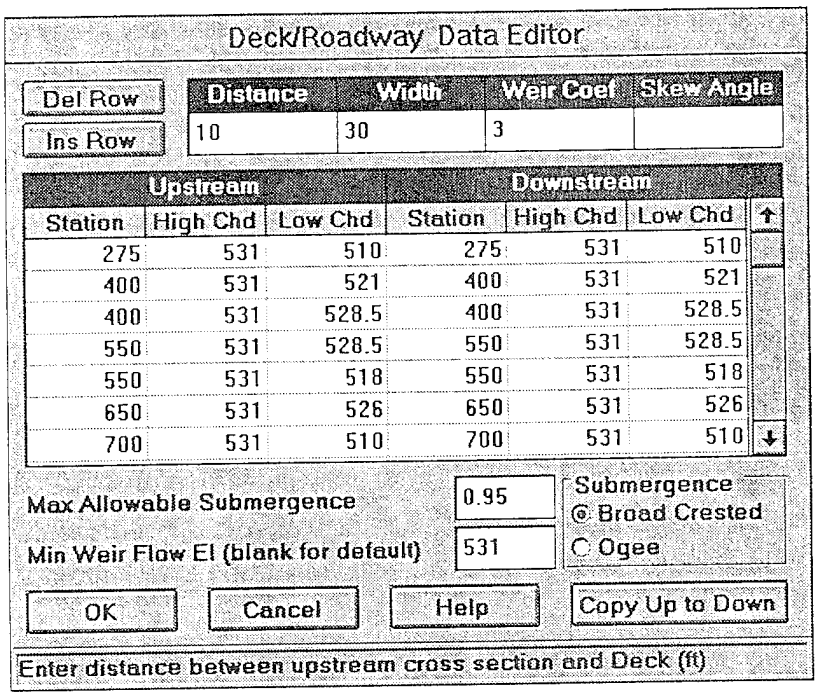


Figure 6.12 Bridge Deck/Roadway Data Editor

Upstream Stationing, High Cord, and Low Cord - This table is used to define the geometry of the bridge deck on the upstream side of the bridge. The information is entered from left to right in cross section stationing. The deck is the area between the high and low chord elevation information. The stationing of the deck does not have to equal the stations in the bounding cross section, but it must be based on the same origin. The **Del Row** and **Ins Row** buttons allow the user to delete and insert rows.

Downstream Stationing, High Cord, and Low Chord - This portion of the table is used to define the geometry of the bridge deck on the downstream side of the bridge. If the geometry of the downstream side is the same as the upstream side, then the user only needs to press the **Copy Up to Down** button. When this button is pressed, all of the upstream bridge deck information is copied to the downstream side. If the bridge deck information on the downstream side is different than the upstream side, then the user must enter the information into the table.

Max Allowable Submergence - The maximum allowable submergence ratio that can occur during weir flow calculations over the bridge deck. If this ratio is exceeded, the program switches to energy based calculations rather than pressure and weir flow.

Submergence Criteria - When submergence occurs there are two choices available to figure out how much the weir coefficient should be reduced due to the submergence. The first method is based on work that was done on a trapezoidal shaped broad crested weir (FHWA, 1978). The second criterion was developed for an Ogee spillway shape (COE,1965). The user should pick the criterion that best matches their problem.

Min Weir Flow El - This field is used to set the minimum elevation of the weir, and therefore the elevation at which weir flow begins. If this field is left blank, the elevation that triggers weir flow is based on the lowest high chord elevation on the upstream side of the bridge deck. If the user enters a value in the field, the value set will be used to determine when weir flow calculations begin. All weir flow calculations are still based on the actual geometry of the deck. Also, weir flow is based on the elevation of the energy grade line and not the water surface.

Once all of the bridge deck information is entered, the user should press the **OK** button at the bottom of the window. Pressing the **OK** button tells the interface to accept the data and close the window. Once the deck editor closes, the graphic of the bridge deck will appear on the Bridge/Culvert Data window. An example of this is shown in Figure 6.13. **Note! The data are not saved to the hard disk at this point.** Geometric data can only be saved to the hard disk from the **File** menu of the Geometric Data window.

Bridge Abutments. The bridge abutments are used to supplement the bridge deck information. Whenever bridge abutments are protruding towards the main channel (sloping inward abutments), it will be necessary to block out additional area that cannot be accounted for in the bridge deck. In this case the user has the ability to add abutments into the data set. To add abutments the user presses the **Abutment** button on the Bridge/Culvert Data editor. Once this button is pressed the Abutment data editor will appear as in Figure 6.14.

Abutments are entered in a similar manner to the bridge deck. When the editor is open, it has already established an abutment # of 1. Generally a left and right abutment are entered for each bridge opening. Abutment data are entered from the abutment left station to the abutment right station. All area below the abutment information is filled in and considered part of the ground. In general it is usually only necessary to enter two points to describe each abutment.

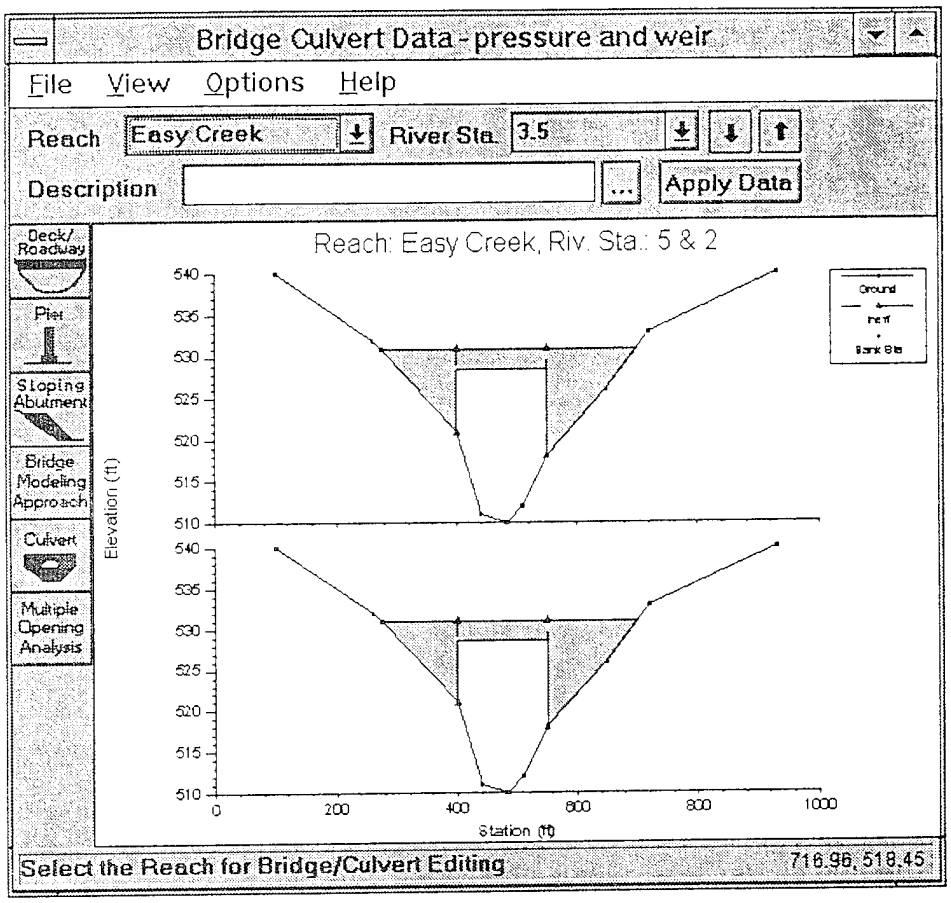


Figure 6.13 Example Bridge Deck Plotted on Bounding Cross Sections

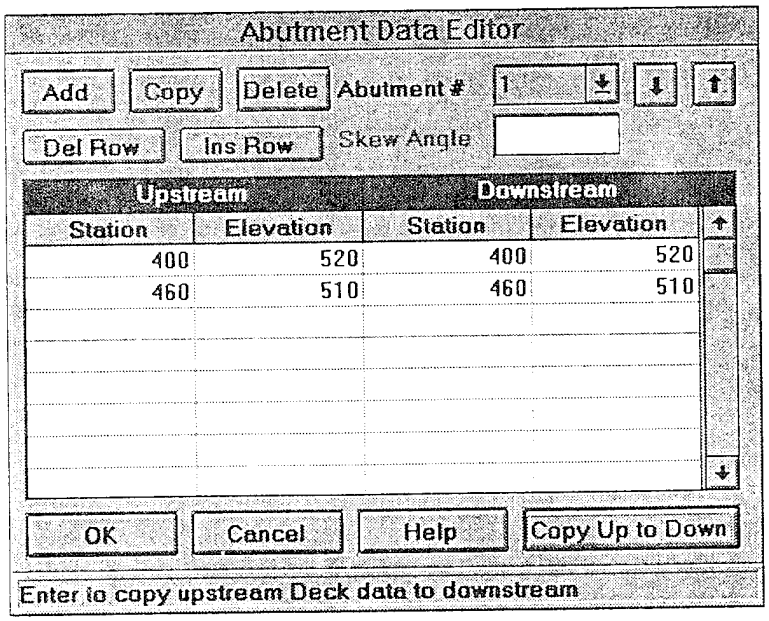


Figure 6.14 Abutment Data editor

The data for each abutment consist of a skew angle (this is optional) and the station and elevation information. The station and elevation information represents the high chord information of the abutment. The low chord information of the abutment is assumed to be below the ground, and it is therefore not necessary to enter it. The geometric information for each abutment can vary from upstream to downstream. If this information is the same, then the user only needs to enter the upstream geometry and then press the **Copy Up to Down** button.

To add additional abutments, the user can either press the **ADD** or the **Copy** button. To delete an abutment, press the **Delete** button. Once all of the abutment data are entered, the user should press the **OK** button. When the **OK** button is pressed, the abutment information is accepted and the editor is closed. The abutments are then added to the bridge graphic on the Bridge/Culvert Data editor.

Bridge Piers. The bridge pier editor is used to describe any piers that exist in the bridge opening. **Note! All piers must be entered through the Pier Editor, they should not be included as part of the ground or bridge deck.** Several of the low flow bridge computations require that the piers be defined separately in order to determine that amount of area under the water surface that is blocked by the piers. If the piers are included with the ground or the bridge deck, several of the methods will not compute the correct amount of energy loss.

To enter pier information, the user presses the **Pier** button on the Bridge/Culvert Data editor. Once the pier button is pressed, the pier data editor will appear as in Figure 6.15 (except yours will not have any data in it yet)

When the pier data editor appears it will have already defined the first pier as pier # 1. The user is required to enter a centerline station for both the upstream and downstream side of the pier. The skew angle is entered in degrees that the pier is skewed from a line parallel to the flow. The skew angle is an optional item. The pier geometry is entered as pier widths and elevations. The elevations must start at the lowest value and go to the highest value. Generally the elevations should start below the ground level. Any pier area below the ground will be clipped off automatically. Pier widths that change at a single elevation are handled by entering two widths at the same elevation. The order of the widths in the table is very important. Keep in mind that the pier is defined from the ground up to the deck. If the pier geometry on the downstream side is the same as the upstream side, simply press the **Copy Up to Down** button after the upstream side data are entered.

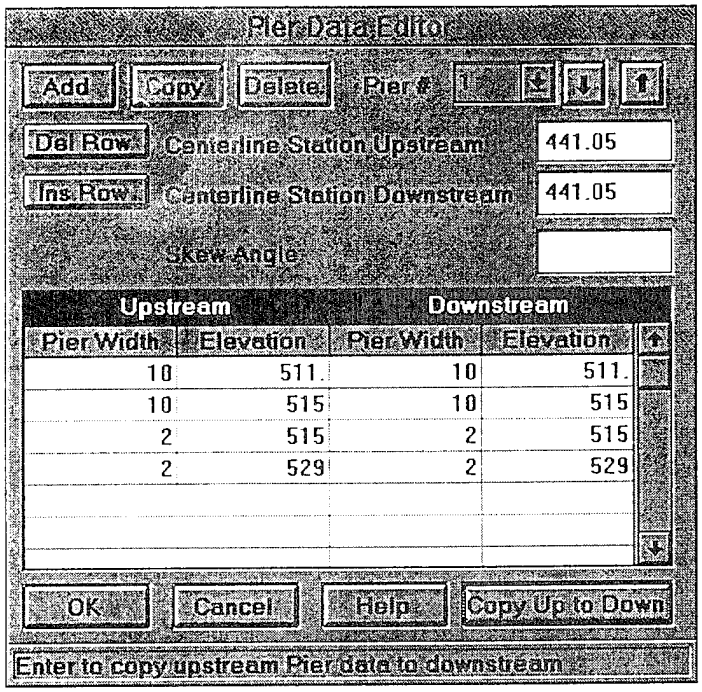


Figure 6.15 Pier Data Editor

Additional piers can be added by pressing either the **Add** or the **Copy** button. If the piers are the same shape, it is easier to use the copy button and simply change the centerline stations of the new pier. To delete a pier, simply press the **Delete** button and the currently displayed pier will be deleted. Once all of the pier data are entered, press the **OK** button. When the OK button is pressed, the data will be accepted and the pier editor will be closed. The graphic of the bridge will then be updated to include the piers. An example bridge with piers is shown in Figure 6.16. This graphic is only the upstream side of the bridge with a zoomed in view.

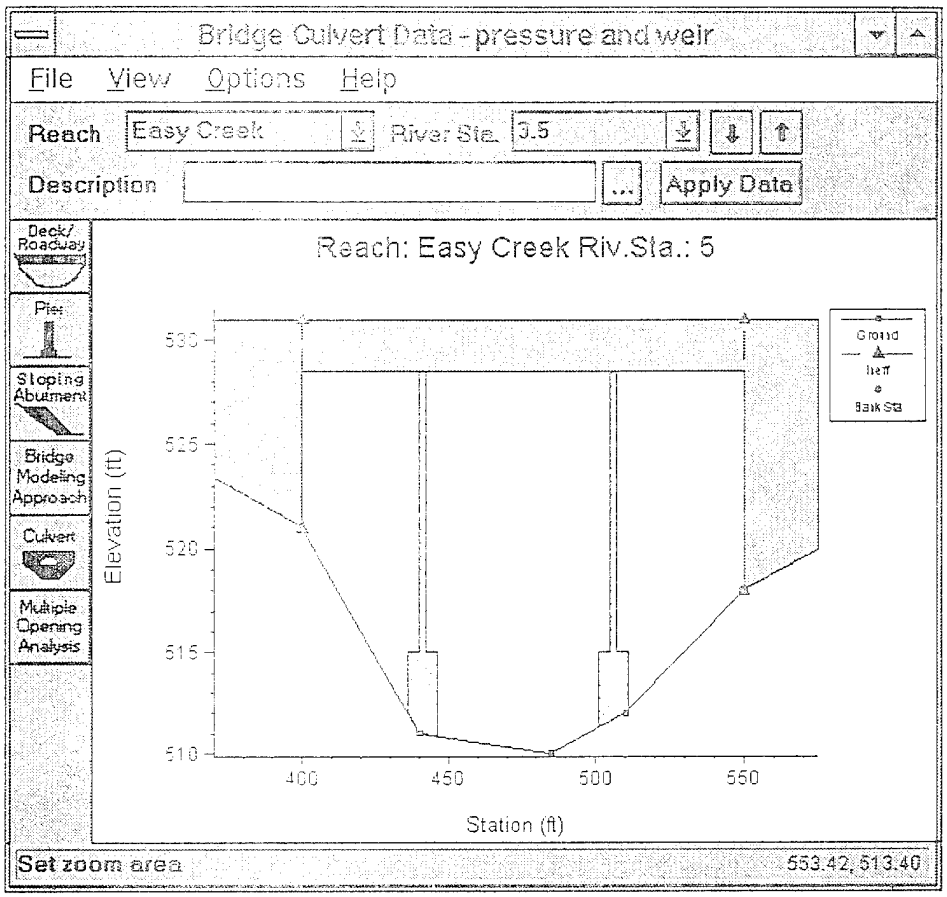


Figure 6.16 Bridge with Piers, zoomed in view

Bridge Modeling Approach. The Bridge Modeling Approach editor is used to define how the bridge will be modeled and to enter any coefficients that are necessary. To bring up the Bridge Modeling Approach editor press the **Bridge Modeling Approach** button on the Bridge/Culvert Data editor. Once this button is pressed, the editor will appear as shown in Figure 6.17 (except yours will only have the default methods selected).

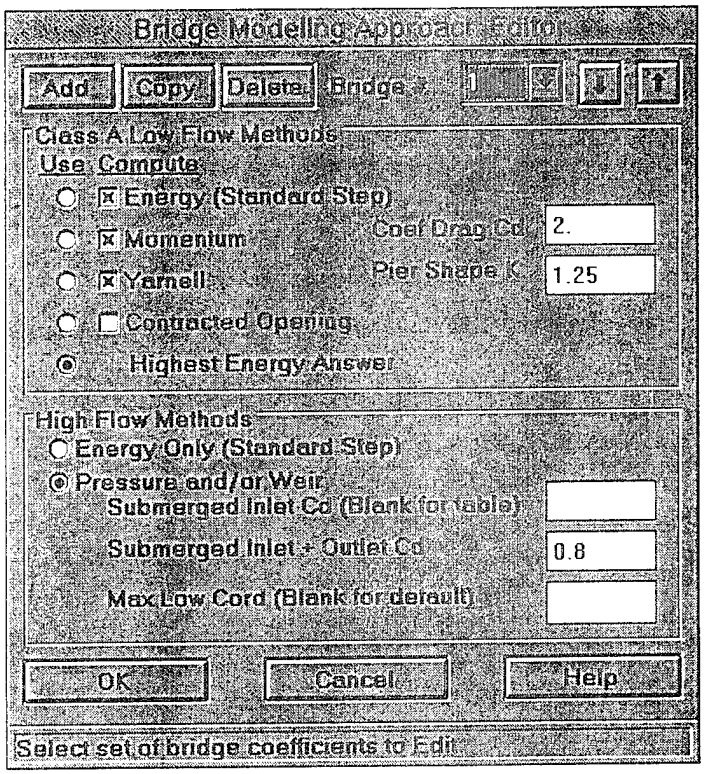


Figure 6.17 Bridge Modeling Approach Editor

When the Bridge Modeling Approach editor comes up it will be ready to enter data for the first bridge opening (coefficient set # 1). If there is more than one bridge opening at the current location, the user can either use a single set of modeling approaches and coefficients, or establish a different set for each bridge opening.

Establishing a bridge modeling approach consists of defining which methods the program will use for class A low flow computations and high flow (flow at or above the maximum low chord) computations. The user can instruct the program to use any or all of the low flow methods during the computations by clicking the buttons under the **Compute** column. If either the Momentum or Yarnell method are selected, the user must enter a value for the coefficient that corresponds to that method. Only one of the methods can be selected as the answer to "**Use**" in order to continue the computations upstream. An alternative to selecting a single method to use is to instruct the program to use the answer with the highest computed upstream energy elevation. This is accomplished by pressing the button under the "**Use**" column that corresponds to the **Highest Energy Answer** text field.

For high flows, the modeler can choose between Energy based calculations or pressure and weir flow calculations. If pressure and weir flow is the selected

high flow method, the user must enter coefficients for the pressure flow equations. The first coefficient applies to the equation that is used when only the upstream side (inlet) of the bridge is submerged. If this coefficient is left blank, the program selects a coefficient based on the amount of submergence. If the user enters a coefficient, then that value is used for all degrees of submergence. The second coefficient applies to the equation that is used when both the upstream and downstream end of the bridge is submerged. Generally this coefficient is around 0.8. For more information on pressure flow coefficients see Hydraulics of Bridge Waterways (FHWA, 1978).

Max Low Chord - This field is used to set the maximum elevation of the deck low chord, and therefore the elevation at which pressure flow begins to be calculated. If this field is left blank, then the elevation that triggers pressure flow calculations is based on the highest low chord elevation on the upstream side of the bridge deck. If the user enters a value in this field, then the value set will be used to trigger when pressure flow calculations begin. Pressure flow is triggered when the energy elevation exceeds the maximum low chord. When pressure flow is calculated, the answer is compared to the low flow answer and the higher of the two is selected. Alternatively, the user can tell the program to use the water surface instead of the energy elevation to trigger pressure flow calculations.

Once all of the bridge modeling approach information is entered, the user should press the OK button. When the OK button is pressed the information will be accepted and the editor will close. **Remember! The data are not saved to disk at this point, it is only accepted as being valid.** To save the geometric data, use the **File** menu from the Geometric Data Editor window.

Culvert Hydraulic Computations

The culvert hydraulic computations in HEC-RAS are similar to the bridge hydraulic computations, except the Federal Highway Administration's (FHWA) standard equations for culvert hydraulics under inlet control are used to compute the losses through the structure. Because of the similarities between culverts and other types of bridges, the cross section layout, the use of ineffective areas, the selection of contraction and expansion coefficients, and many other aspects of bridge analysis apply to culverts as well.

The culvert routines in HEC-RAS have the ability to model six different types of culvert shapes. These shapes include box (rectangular), circular, elliptical, arch, pipe arch, and semi circular culverts. The culvert routines are limited to subcritical and mixed flow regime water surface profiles. If you are running a supercritical only profile, when the model encounters a culvert, critical depth is calculated inside the culvert and used as the water surface. The computations then continue downstream starting with a critical depth boundary condition at the downstream side of the culvert.

The analysis of flow in culverts is complicated. It is common to use the concepts of "Inlet" control and "Outlet" control to simplify the analysis. **Inlet control** flow occurs when the flow carrying capacity of the culvert entrance is less than the flow capacity of the culvert barrel. **Outlet control** flow occurs when the culvert carrying capacity is limited by downstream conditions or by the flow capacity of the culvert barrel. The HEC-RAS culvert routines compute the headwater required to produce a given flow rate through the culvert for inlet control conditions and for outlet control conditions. The higher headwater "controls", and an upstream water surface is computed to correspond to that energy elevation.

Inlet Control Computations. For inlet control, the required headwater is computed by assuming that the culvert inlet acts as an orifice or a weir. Therefore, the inlet control capacity depends primarily on the geometry of the culvert entrance. Extensive laboratory tests by the National Bureau of Standards, and the Bureau of Public Roads (now, FHWA), and other entities resulted in a series of equations which describe the inlet control headwater under various conditions. These equations are used by HEC-RAS in computing the headwater associated with inlet control.

Outlet Control Computations. For outlet control flow, the required headwater must be computed considering several conditions within the culvert and the downstream tailwater. For culverts flowing full, the total energy loss through the culvert is computed as the sum of friction losses, entrance losses, and exit losses. Friction losses are based on Manning's equation. Entrance losses are computed as a coefficient times the velocity head in the culvert at the upstream end. Exit losses are computed as a coefficient times the change in velocity head from just inside the culvert (at the downstream end) to outside the culvert.

When the culvert is not flowing full, the direct step backwater procedure is used to calculate the profile through the culvert up to the culvert inlet. An entrance loss is then computed and added to the energy inside the culvert (at the upstream end) to obtain the upstream energy (headwater). For more information on the hydraulics of culverts, the reader is referred to the HEC-RAS Hydraulics Reference manual.

Entering and Editing Culvert Data

Culvert data are entered in the same manner as bridge data. To enter culvert data the user presses the **Bridge/Culvert** button on the Geometric Data window (Figure 6.1). Once this button is pressed, the Bridge/Culvert Data Editor will appear (Figure 6.11). To add a culvert group to the model the user must then do the following:

1. Select the reach that you would like to place the culvert in. This is accomplished by pressing the down arrow on the reach box, and then selecting the reach of choice.

2. Go to the **Options** menu of the Bridge/Culvert editor and select **Add a Bridge and/or Culvert** from the list. An input box will appear prompting you to enter a river station identifier for the new culvert group. After entering the river station, press the **OK** button and the cross sections that bound the new culvert group will appear in the editor.

3. Enter all of the required data for the culvert group. This includes the culvert deck information and the culvert specific data. The culvert deck and roadway information is entered in the same manner as a bridge (using the deck editor). To enter culvert specific data, press the **Culvert** button on the Bridge/Culvert Data editor.

4. Once all of the culvert data are entered, press the **OK** button in order for the interface to accept the information.

Reach and River Station. The Reach box allows the user to select a reach from the available reaches that were put together in the schematic diagram. The reach label defines which reach the culvert will be located in. The River Station tag defines where the culvert will be located within the specified reach. The River Station tag does not have to be the actual river station of the culvert, but it must be a numeric value. The River Station tag for the culvert should be numerically between the two cross sections that bound the culvert. Once the user selects **Add a Bridge and/or Culvert** from the options menu, an input box will appear prompting you to enter a River Station tag for the new culvert. After the River Station tag is entered, the two cross sections that bound the culvert will be displayed on the editor.

Description. The description box is used to describe the culvert location in more detail than just the reach and river station. This box has a limit of 256 characters. Only the first line of information is displayed, unless the button to the right of the box is pressed. Also, the first 40 characters of the description are used as a label for culvert plots and tables.

Culvert Deck. The culvert deck is virtually the same as the bridge deck/roadway information. The deck is used to describe the area blocking the stream and the roadway profile. The only difference in the deck information for culverts is that the low chord elevations should be left blank or set to elevations below the ground data. This will cause the deck to completely fill the channel up to the roadway elevations (high chord data). Therefore, the only opening below the roadway will be whatever culvert openings are entered.

To enter the culvert deck information, press the **Deck** button on the Bridge/Culvert Data Editor window. For an explanation of the deck information, please review the section entitled **Bridge Deck** found earlier in this chapter.

Culvert Data. To enter culvert specific information, press the **Culvert** button on the Bridge/Culvert Data Editor window. When this button is pressed, the Culvert Data Editor will appear as shown in Figure 6.18 (except yours will be blank). The information entered in the Culvert Data Editor consists of the following:

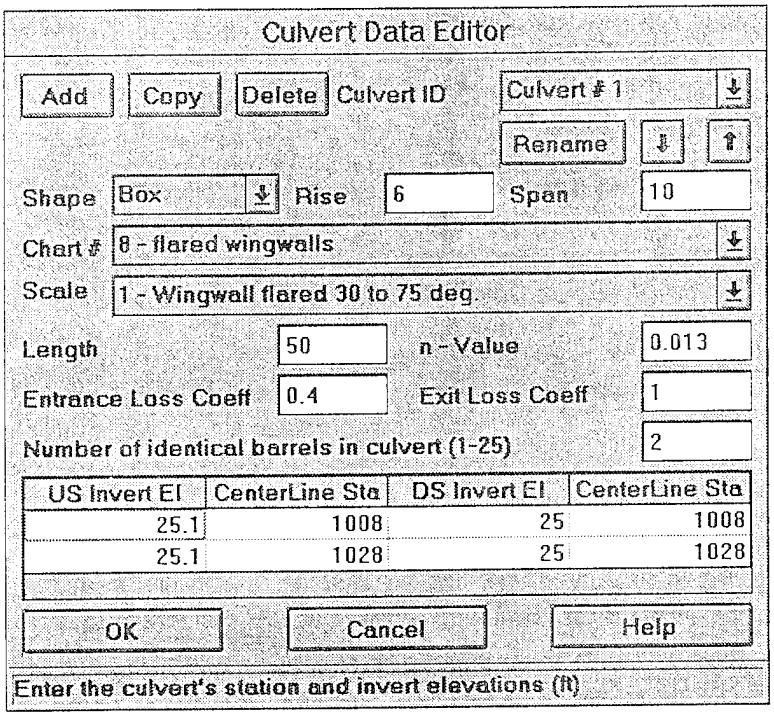


Figure 6.18 Culvert Data Editor

Culvert ID# - The culvert identifier (ID#) is automatically assigned to "Culvert #1" the first time you open the editor. The user can enter more than one culvert type if they are working on a multiple culvert opening problem. If all of the culvert barrels are exactly the same, then only one culvert type (Culvert ID#) should be entered. The number of barrels is an input parameter in the culvert data. If the user has culverts that are different in shape, size, elevation, or loss coefficients, then additional culverts types (Culvert ID#'s) must be added for each culvert type. To add an additional culvert type you can either use the **Add** or **Copy** buttons. The Add button increments the culvert ID# and clears the culvert editor. The Copy button increments the culvert ID# and makes a copy of the original culvert data. Once a copy is made of a culvert, the user can change any of the existing culvert information. Culverts can be deleted by pressing the **Delete** button.

Shape - The shape selection box allows the user to select from one of the six available shapes. This is accomplished by pressing the down arrow on the side of the box, and then selecting one of the six available shapes.

Rise - The rise field describes the maximum height inside of the culvert.

Span - The span field is used to define the maximum width inside of the culvert. The span is left blank for circular culverts.

Chart #. - This field is used to select the Federal Highway Administration Chart number that corresponds to the type and shape of culvert being modeled. Once the user has selected a culvert shape, the corresponding FHWA chart numbers will show up in the chart # selection box. More information on FHWA chart numbers can be found in the Hydraulics Reference manual.

Scale. - This field is used to select the Federal Highway Administration Scale number that corresponds to the type of culvert entrance. Once the user has selected a culvert shape and chart #, the corresponding FHWA scale numbers will show up in the scale selection box. More information on FHWA scale numbers can be found in the Hydraulics Reference manual.

Length - The length field describes the length of the culvert along the centerline of the barrel.

n-value - The n-value field is used for entering the Manning's n value of the culvert barrel.

Entrance Loss Coefficient - The coefficient entered in this field will be multiplied by the velocity head inside the culvert at the upstream end. This value represents the amount of energy loss that occurs as flow enters the culvert.

Exit Loss Coefficient - The coefficient entered in this field will be multiplied by the change in velocity head from inside the culvert to outside the culvert at the downstream end. This value represents the energy loss that occurs as water exits the culvert.

Number of Identical Barrels in Culvert - This field is used to enter the number of identical barrels. The number of identical barrels is limited 25. However, if more than one barrel is entered, the user must provide different centerline stationing information for each barrel.

Upstream Invert Elevation and Centerline Station - These two fields are used to describe where the culvert is located in reference to the cross section on the upstream side.

Downstream Invert Elevation and Centerline Station - These two fields are used to describe where the culvert is located in reference to the cross section on the downstream side.

Once all of the culvert information is entered, the user should press the **OK** button at the bottom of the window. Pressing the **OK** button tells the interface to accept the data and close the window. Once the culvert editor is closed, the graphic of the culvert will appear on the Bridge/Culvert Data editor window. An example culvert with two identical barrels is shown in Figure 6.19. **Note! The data are not saved to the hard disk at this point.** Geometric data can only be saved from the **File** menu on the Geometric Data window.

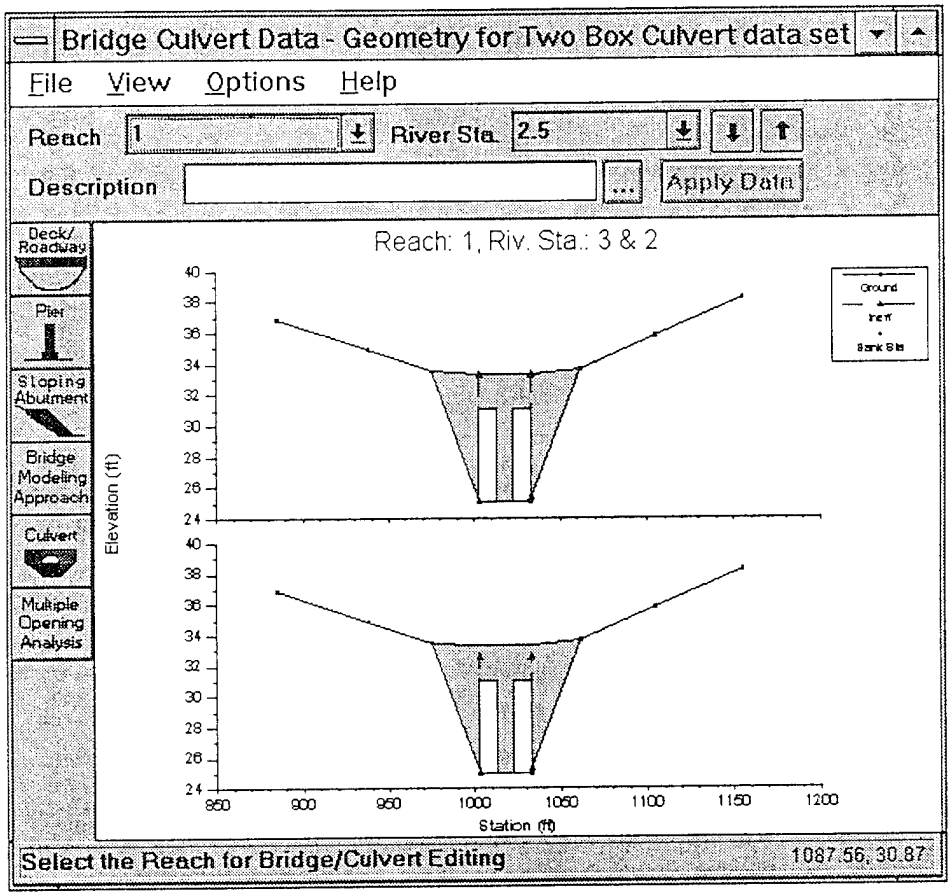


Figure 6.19 Bridge/Culvert Data Editor with example culvert

Bridge and Culvert Options

Some additional options that are available, but not required, are found under the **Options** menu from the Bridge/Culvert Data Editor. These include the following:

Add a Bridge and/or Culvert. This option initiates the process of adding a bridge or culvert to the data set. The user is prompted to enter a river station tag for the new bridge or culvert. The river station tag locates the bridge or culvert within the selected reach. Once the river station is entered, the

Bridge/Culvert Data editor is cleared and the user can begin entering the data for that new bridge or culvert.

Delete Bridge and/or Culvert. This option will delete the currently displayed bridge or culvert. The user is prompted with a message stating specifically which bridge or culvert is going to be deleted, and requesting them to press the **OK** button or the **Cancel** button.

Ineffective Flow Areas. This option allows the user to establish ineffective flow areas at the cross sections just upstream and downstream of the bridge. Ineffective flow areas defined in the bridge/culvert data editor are exactly the same as in the cross section data editor. They are only provided here for convenience. For more information on ineffective flow areas, go to the ineffective flow area section under cross section options.

Momentum Equation. This option allows the user to change the components of the momentum equation. The momentum equation is one of the optional low flow methods in the bridge routines. The default momentum equation includes terms in the equation to account for friction losses and the weight of water component. The user can turn either or both of these components off from this option.

Momentum Class B Defaults. If the program computes that the flow must pass through critical depth inside the bridge (Class B flow), critical depth will automatically be located inside the bridge at the upstream end. This option allows the user to control where the program sets critical depth for class B flow. If the user feels that it would be better to set critical depth inside the bridge at the downstream end, then this can be selected.

Pressure Flow Criteria. This option allows the user to select either the energy grade line or the water surface, to be used as the criterion for when the program begins checking for the possibility of pressure flow. By default the program uses the energy grade line.

Bridge and Culvert View Features

Several options are available for viewing the bridge/culvert geometric data. These options include: Zoom In; Zoom Out; Display Upstream XS; Display Downstream XS; Display Both; Highlight Weir, Opening Lid and Ground; Highlight Piers; and Grid. These options are available from the **View** menu on the bridge/culvert data editor.

Zoom In: This option allows the user to zoom in on a piece of the bridge or culvert. This is accomplished by selecting **Zoom In** from the **View** menu, then specifying the area to zoom in on with the mouse. Defining the zoom area is accomplished by placing the mouse pointer in the upper left corner of the desired area. Then press down on the left mouse button and drag the

mouse to define a box containing the desired zoom area. Finally, release the left mouse button and the viewing area will display the zoomed in area of the bridge or culvert.

Zoom Out: This option re-displays the bridge or culvert back into its original size before you zoomed in. Zooming out is accomplished by selecting **Zoom Out** from the **View** menu bar on the bridge/culvert data editor.

Display Upstream XS. When this option is selected, only the upstream side of the bridge or culvert will be displayed.

Display Downstream XS. When this option is selected, only the downstream side of the bridge or culvert will be displayed.

Display Both. When this option is selected, both the downstream and upstream sides of the bridge will be displayed in the viewing area.

Highlight Weir, Opening Lid and Ground. When this option is selected, various portions of the bridge/culvert graphic will be highlighted. This allows the user to see exactly what the program is going to use as the weir, the opening lid (the top of the bridge opening), and the opening ground (bottom of the bridge opening). This option is very useful for detecting any data entry errors that may otherwise go unnoticed.

Highlight Piers. When this option is turned on the interface will highlight what it thinks is the extent of the pier information. This option allows the user to see exactly what the program thinks are piers and to see how the pier information has been clipped. Piers are clipped below the ground and above the low chord of the bridge.

Grid. This option allows the user to have a grid overlaid on top of the bridge or culvert graphic.

Multiple Bridge and/or Culvert Openings

HEC-RAS has the ability to model multiple bridge and/or culvert openings at any individual river crossing. Types of openings can consist of bridges, culvert groups (a group of culverts is considered to be a single opening), and conveyance areas (an area where water will flow as open channel flow, other than a bridge or culvert opening). Up to seven openings can be modeled at a given location, and any combination of bridges and culvert groups can be used. Conveyance type openings can range from zero to a maximum of two.

An example multiple opening is shown in Figure 6.20. As shown in this example, there are three types of openings: a conveyance area (left side, labeled as opening #1), a bridge (labeled as opening #2), and a culvert group (labeled as opening #3). During low flow conditions, flow will be limited to

the bridge opening. As flow increases, the culverts will begin to take some of the flow away from the bridge opening. The conveyance area was defined as ineffective flow (no conveyance) until the water surface goes above the top of the bridge. This was accomplished by setting blocked ineffective flow areas. In this example, three blocked ineffective flow areas were established: one to the left of the bridge (which encompasses the whole conveyance area), one between the bridge and the culvert group, and one to the right of the culvert group.

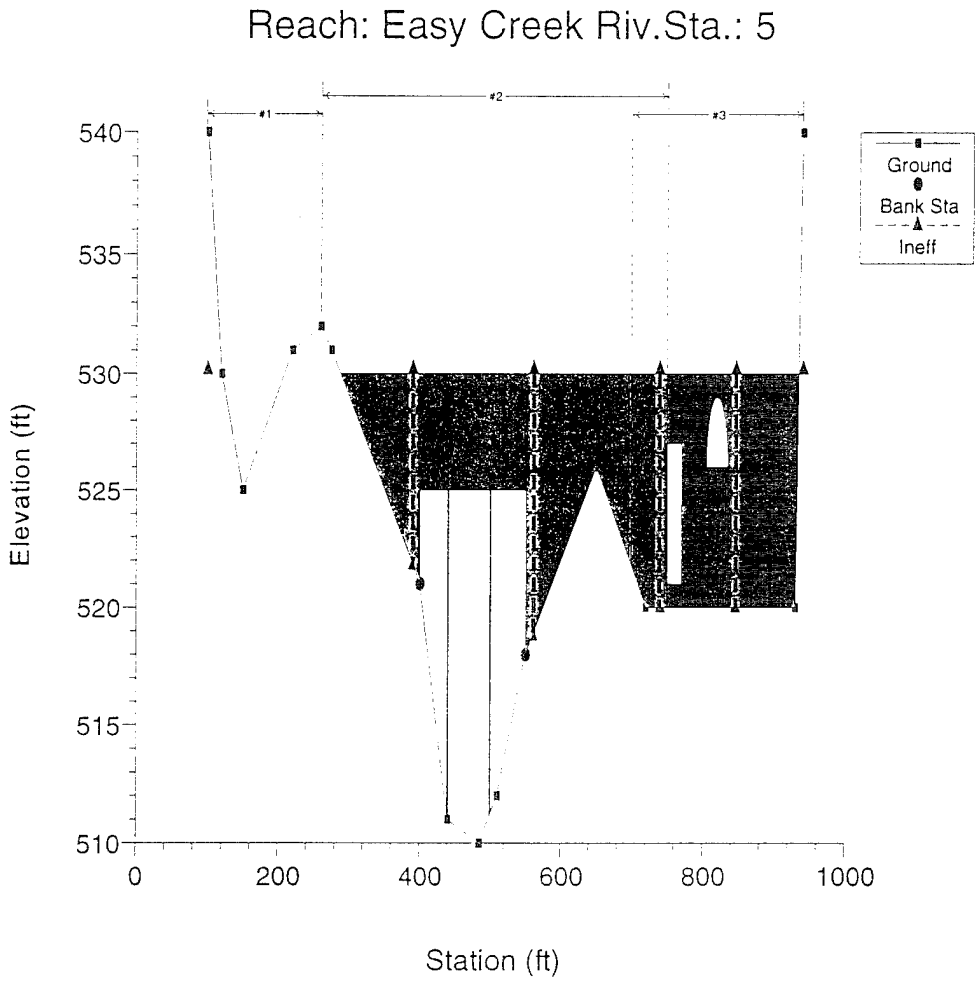


Figure 6.20 Example Multiple Opening River Crossing

Entering Multiple Opening Data

Multiple opening data are entered in the same manner as any other bridge or culvert crossing. In general, the user should perform the following steps to enter multiple opening data:

1. Press the Bridge\Culvert button on the Geometric Data window.

2. Select the reach in which you would like to place the multiple opening river crossing. This is accomplished from the Reach box near the top of the window.

3. Enter the river station at which you want to place the multiple opening crossing. Once you have done this, the two cross sections that bound this river station will appear in the window. These two cross sections, along with the bridge and culvert information, will be used to formulate the two cross sections inside the multiple opening river crossing.

4. Enter the deck and road embankment data by using the Deck editor.

5. Enter any piers or abutments that are required.

6. Select the **Bridge Modeling Approach** button and enter a set of coefficients and modeling approaches for each bridge opening.

7. Enter Culvert data for any culvert openings.

8. Select the **Multiple Opening Analysis** button on the bridge and culvert editor. Enter the types of openings and their station limits. Start at the left most station of the crossing and work your way to the right end. This is explained in greater detail under the section entitled "Defining the Openings".

Deck/Road Embankment Data. There can only be one deck and road embankment entered for any bridge and\or culvert crossing. The deck editor is used to describe the area that will be blocked out due to the bridge deck and road embankment. As shown by the grey shaded area in Figure 6.20, the deck and roadway data are used to block out area around the bridge as well as around the culverts. In the area of the bridge, high and low chord information is entered in order to define the top of road as well as the bridge opening. In the area of the culverts, the high chord information is entered to define the rest of the top of the road embankment. However, the low chord information can be left blank, or set to elevations below the ground, because the culvert data define the culvert openings.

Piers and abutments. All piers are entered from the pier editor, which was described previously under bridge data. The number of bridge openings has no impact on how pier data are entered. Piers are treated as separate information. Once the user establishes that there is more than one bridge opening, the program is smart enough to figure out which piers go with which opening. If any abutment data are required for a bridge opening, it can be entered as described previously under the bridge data section.

Bridge Modeling Approach. A bridge modeling approach and coefficient set must be established for at least one bridge opening. If there is more than one bridge opening, and the user has only established a single coefficient set and bridge modeling approach, those data will be used for all of the bridge openings. The user can establish a different set of coefficients and modeling approaches for each bridge opening.

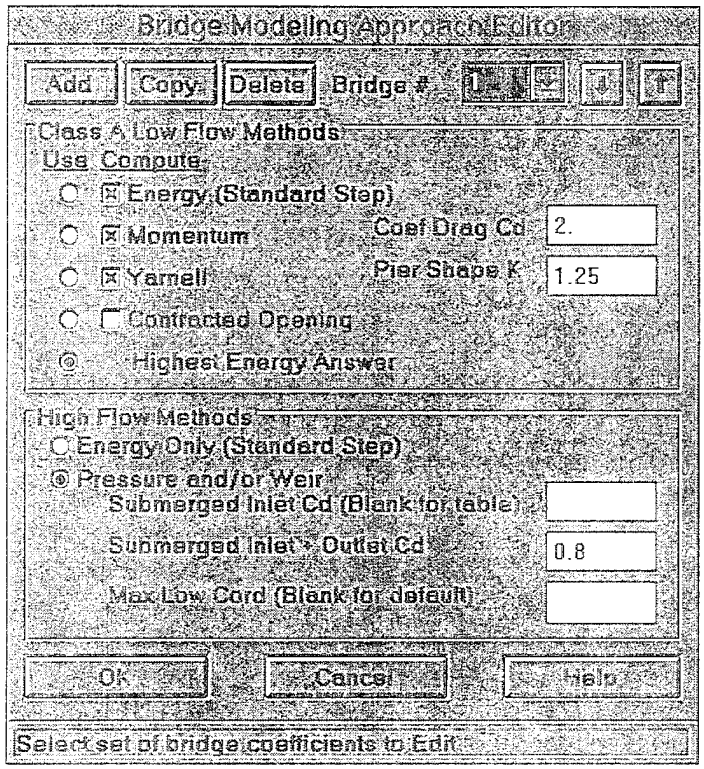


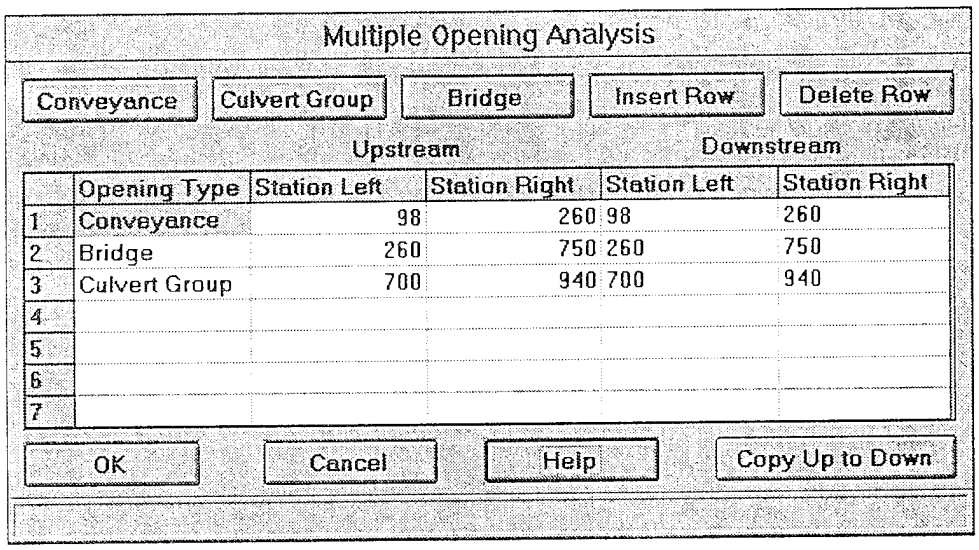
Figure 6.21 Bridge Modeling Approach Editor

As shown in Figure 6.21, the user must enter information under the Bridge Modeling Approach editor for at least one bridge Opening. Bridge openings are referred to as Bridge # 1, Bridge # 2, etc.., up to the number of bridge openings. Bridge # 1 represents the left most bridge opening while looking in the downstream direction. Bridge # 2 represents the next bridge opening to the right of Bridge # 1, and so on. The user can enter additional coefficient sets and modeling approaches by selecting either the add or copy button. If either of these buttons is selected, the Bridge number will automatically be incremented by one. The user can then enter or change any of the information on the editor for the second bridge opening. Any bridge opening that does not have a corresponding coefficient set and modeling approach, will automatically default to what is set for Bridge # 1.

Culvert Data. Culvert information is added in the same manner as described in the previous section called "Entering and Editing Culvert Data." Culverts will automatically be grouped based on their stationing.

Defining The Openings

Once all of the bridge and\or culvert data are entered for a multiple opening river crossing, the last step is to define the number and type of openings that are being modeled. This is accomplished by pressing the **Multiple Opening Analysis** button on the Bridge\Culvert Data editor. Once this button is pressed, an editor will appear as shown in Figure 6.22 (except yours will be blank the first time you bring it up).



	Opening Type	Station Left	Station Right	Station Left	Station Right
1	Conveyance	98	260	98	260
2	Bridge	260	750	260	750
3	Culvert Group	700	940	700	940
4					
5					
6					
7					

Figure 6.22 Multiple Opening Analysis window

The user selects from the three available opening types: Conveyance; Culvert Group; and Bridge. Openings must be established in order from left to right, while looking in the downstream direction. In addition to establishing the number and types of openings, the user must also enter a Station Left and a Station Right for each opening. These stations are used to establish limits for each opening as well as stagnation points. Stagnation points are the locations at which flow separates (on the upstream side) from one opening to the next adjacent opening. Stagnation points can either be set to fixed locations or they can be allowed to migrate within limits.

As shown in Figure 6.22 (numerical representation) and Figure 6.20 (graphical representation), there are three openings established in this example. The first opening is defined as a conveyance area, and it ranges from station 98 (the left most station of the section) to station 260. That means that any water in this area will be treated as normal open channel flow,

and the water surface through this area will be solved by performing standard step calculations with the energy equation. The second opening is the bridge opening. This opening has a left station of 260 and a right station of 750. This bridge will be modeled by using the cross section data, bridge deck, and pier information that lie within these two stations (260 and 750). The bridge coefficients and modeling approach for this opening will be based on the data entered for bridge opening #1, since it is the first bridge opening. The third opening is a culvert group. This opening has a left station of 700 and a right station of 940. Any culverts that lie within these stations will be considered as being in the same culvert group.

Notice that the right station of the bridge opening overlaps with the left station of the culvert group. This is done on purpose. By overlapping these stations, the user is allowing the program to calculate the location of the stagnation point between these two openings. This allows the stagnation point to vary from one profile to the next. In the current version of the HEC-RAS software, stagnation points are allowed to migrate between any bridge and culvert group openings. However, stagnation points must be set to a fixed location for any conveyance opening type. A more detailed explanation of stagnation points, and how the program uses them, can be found in the HEC-RAS Hydraulics Reference manual, under the section on Multiple Openings.

Once the user has entered all of the information into the Multiple Opening Analysis window, simply press the **OK** button to accept the data.

Multiple Opening Calculations

Multiple opening calculations are computationally intensive. An iterative solution approach is used, by which the amount of flow through each opening is adjusted until the computed upstream energies of each opening are balanced within a predefined tolerance. The general approach of the solution scheme is as follows:

1. The program makes a first guess at the upstream water surface by setting it to the computed energy of the cross section just downstream of the bridge.

2. The program sets an initial flow distribution. This is accomplished by first calculating the amount of active flow area in each opening, based on the water surface from step one. The program then apportions the flow by using an area weighting (i.e. if an opening has 40 percent of the active flow area, then it will receive 40 percent of the flow).

3. Once a flow distribution is established, the program then calculates the water surface and energy profiles for each opening, using the estimated flow.

4. Once the program has computed the upstream energy for each opening, a comparison is made between the energies to see if a balance has been achieved (i.e., all energies are within the predefined tolerance). If the energies are not within the set tolerance, the program re-distributes the flow based on the computed energies.

5. The program continues this process until either the computed energies are within the tolerance or the number of iterations reaches a pre-defined maximum. The energy balance tolerance is set as 3 times the user entered water surface calculation tolerance (The default is 0.03 feet or 0.009 meters). The maximum number of iterations for multiple opening analysis is set to 1.5 times the user entered maximum number of iterations from the normal water surface calculations (the default is 30 for multiple openings).

A more detailed discussion of how the program performs the multiple opening analysis can be found in the HEC-RAS Hydraulic Reference manual.

Cross Section Interpolation

Occasionally it is necessary to supplement surveyed cross section data by interpolating cross sections in between two surveyed sections. Interpolated cross sections are often required when the change in velocity head is to large to accurately determine the energy gradient. An adequate depiction of the change in energy gradient is necessary to accurately model friction losses as well as contraction and expansion losses.

Cross section interpolation can be accomplished in three ways from within the HEC-RAS interface. The first method is to simply copy one of the bounding cross sections and then adjust the station and elevation data. The cross section editor allows the user to raise or lower elevations and to shrink or expand various portions of any cross section.

The second and third options allow for automatic interpolation of cross section data. From the Geometric Data editor, automatic interpolation options are found under the **Options** menu bar as shown in Figure 6.23.

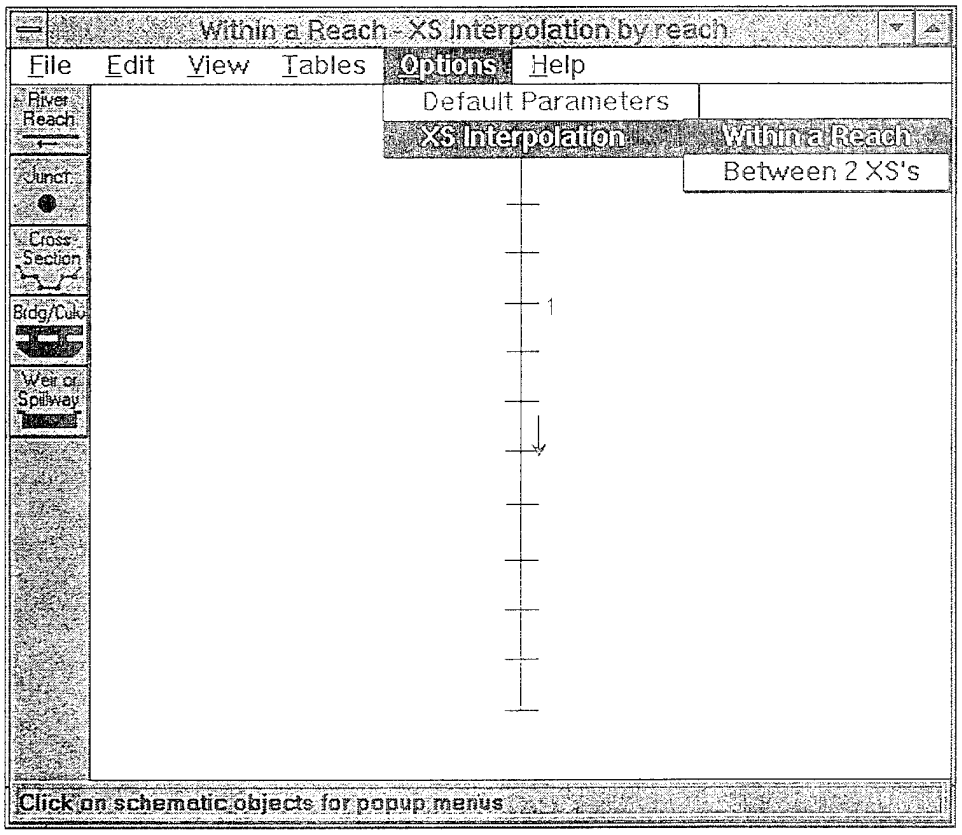


Figure 6.23 Automatic Cross Section Interpolation Options

The first cross section interpolation option, **Within a Reach**, allows for automatic interpolation over a specified range of cross sections within a single reach. When this option is selected, a window will pop up as shown in Figure 6.24. The user must enter a starting River Station and an ending River Station for which interpolation will be performed. The user must also provide the maximum allowable distance between cross sections. If the main channel distance between two sections is greater than the user defined maximum allowable, then the program will interpolate cross sections between these two sections. The program will interpolate as many cross sections as necessary in order to get the distance between cross sections below the maximum allowable.

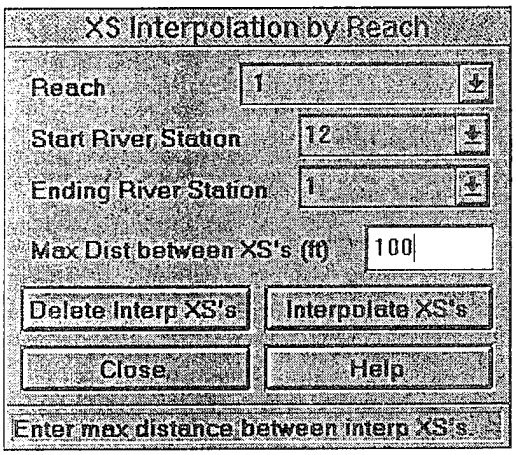


Figure 6.24 Automatic Cross Section Interpolating Within a Reach

Once the user has selected the cross section range and entered the maximum allowable distance, cross section interpolation is performed by pressing the **Interpolate XS's** button. When the program has finished interpolating the cross sections, the user can close the window by pressing the **Close** button. Once this window is closed, the interpolated cross sections will show up on the river schematic as light green tic marks. The lighter color is used to distinguish interpolated cross sections from user entered data. Interpolated cross sections can be plotted and edited like any other cross section. The only difference between interpolated sections and user defined sections is that interpolated sections will have an asterics (*) attached to the end of their river station identifier. This asterics will show up on all input and output forms, enabling the user to easily recognize which cross sections are interpolated and which are user defined.

The second type of automatic cross section interpolation, **Between 2 XS's**, allows the user to have much greater control over how the interpolation is performed. When this option is selected, a Cross Section Interpolation window will appear as shown in Figure 6.25.

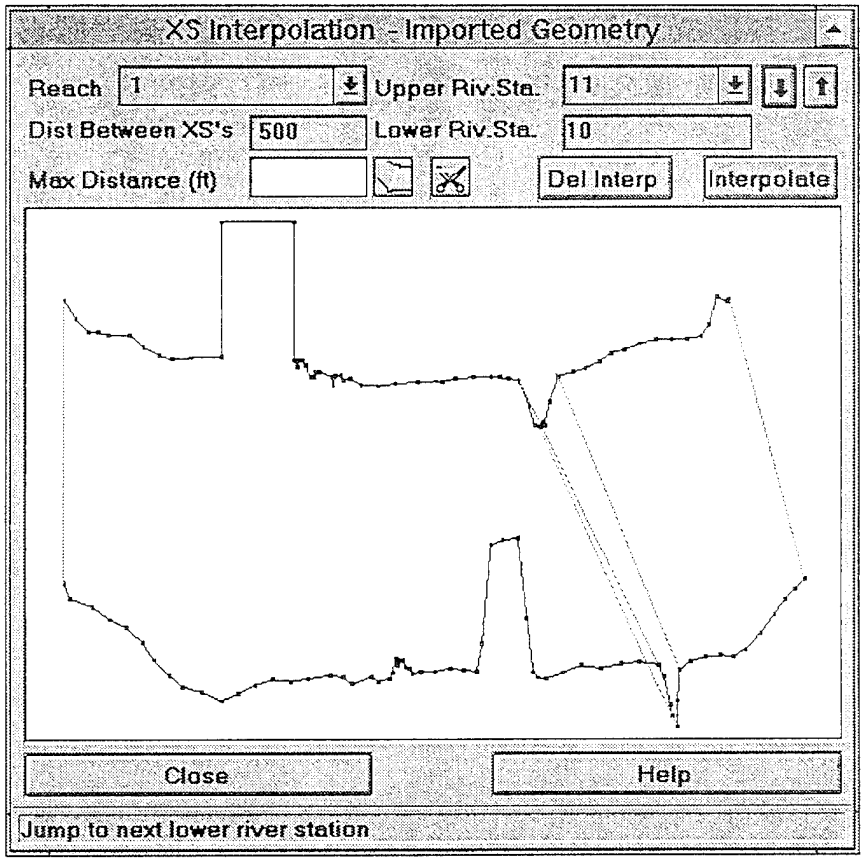


Figure 6.25 Detailed Cross Section Interpolation Window

This cross section interpolation window displays only two cross sections at a time. The user can get to any reach or cross section from the Reach and River Station boxes at the top of the window. Interpolated cross section geometry is based on a string model as graphically depicted in Figure 6.25. The string model consists of chords that connect the coordinates of the upstream and downstream cross sections. The cords are classified as master and minor cords. As shown in Figure 6.25, five master cords are automatically attached between the two cross sections. These master cords are attached at the ends of the cross sections, the main channel bank stations, and the main channel inverts. Minor cords are generated automatically by the interpolation routines. A minor cord is generated by taking an existing coordinate in either the upstream or downstream section and establishing a corresponding coordinate at the opposite cross section by either matching an existing coordinate or interpolating one. The station value at the opposite cross section is determined by computing the decimal percent that the known coordinate represents of the distance between master cords and then applying that percentage to the opposite cross section master cords. The number of minor cords will be equal to the sum of all the coordinates of the upstream and downstream sections minus the number of master cords. Interpolation at any

point in between the two sections is then based on linear interpolation of the elevations at the ends of the master and minor cords. Interpolated cross sections will have station and elevation points equal to the number of major and minor cords.

This interpolation scheme is used in both of the automated interpolation options ("Within a Reach" and "Between 2 XS's"). The difference is that the **Between 2 XS's** option allows the user to define additional master cords. This can provide for a better interpolation, especially when the default of five major cords produces an inadequate interpolation. An example of an inadequate interpolation when using the default cords is shown in Figure 6.26.

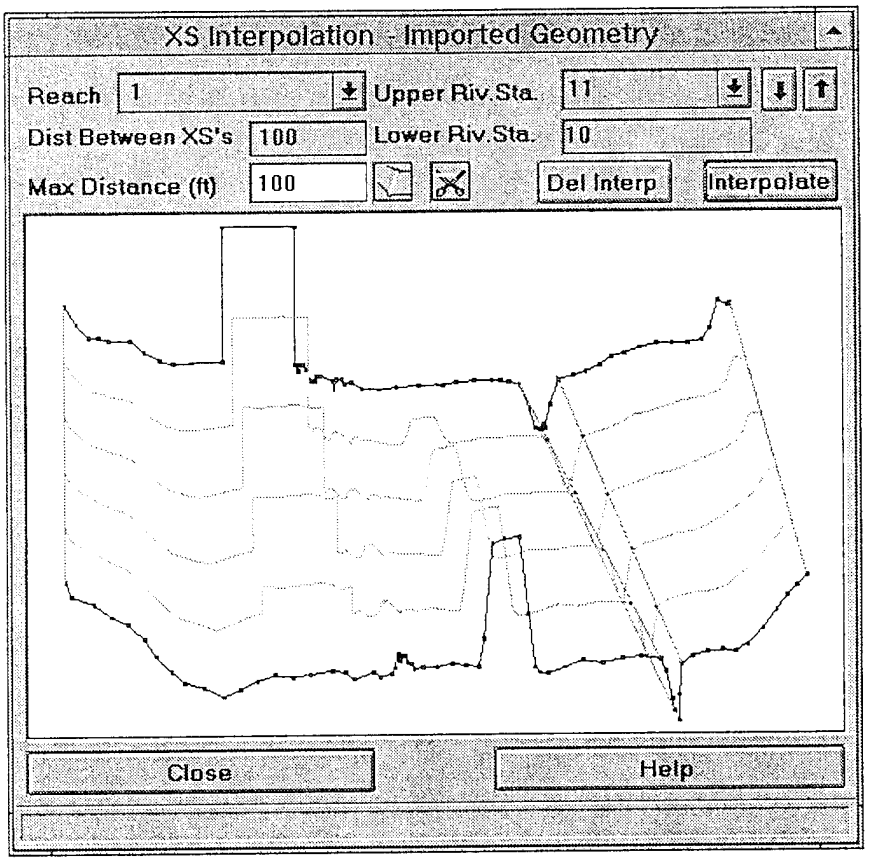


Figure 6.26 Cross Section Interpolation Based on Default Master Cords

As shown in Figure 6.26, the interpolation was adequate for the main channel and the left overbank area. The interpolation in the right overbank area failed to connect two geometric features that could be representing a levee or some other type of high ground. If it is known that these two areas of high ground should be connected, then the interpolation between these two sections should be deleted, and additional master cords can be added to connect the two features. To delete the interpolated sections, press the **Del Interp** button.

Master cords are added by pressing the **Master Cord** button that is located to the right of the Maximum Distance field above the graphic. Once this button is pressed, any number of master cords can be drawn in. Master cords are drawn by placing the mouse pointer over the desired location on the top cross section. Then while holding the left mouse button down, drag the mouse pointer to the desired location of the lower cross section. When the left mouse button is released, a cord is automatically attached to the closest point near the pointer. An example of how to connect master cords is shown in Figure 6.27.

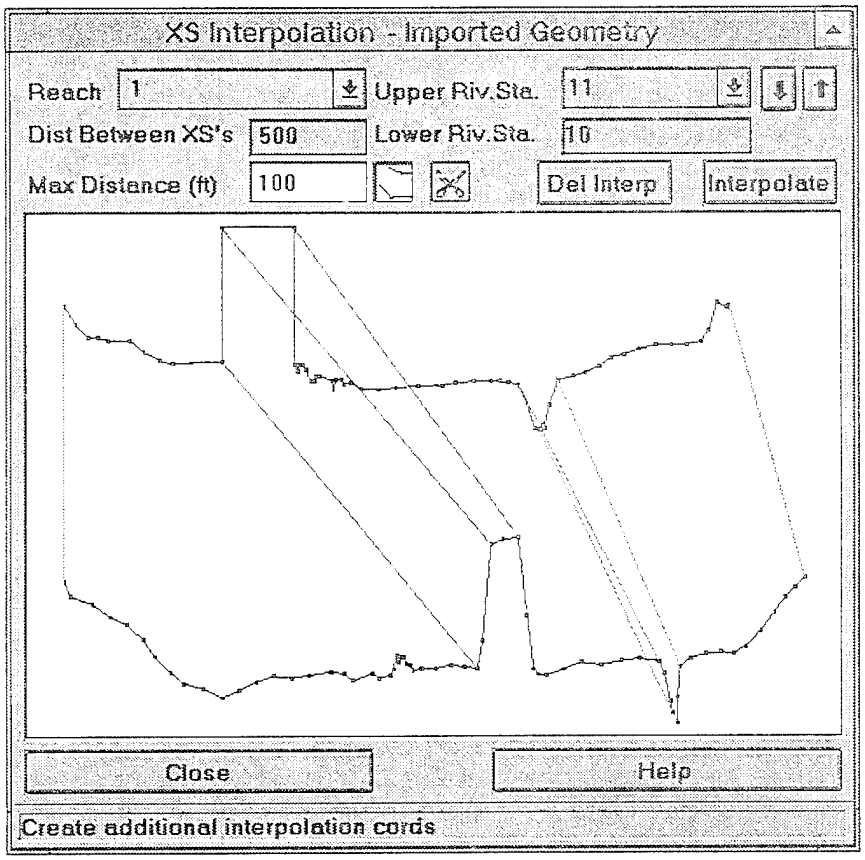


Figure 6.27 Adding Additional Master Cords for Interpolation

User defined master cords can also be deleted. To delete user defined master cords, press the **scissors** button to the right of the master cords button. When this button is pressed, simply move the mouse pointer over a user defined cord and click the left mouse button to delete the cord.

Once you have drawn in all the master cords that you feel are required, and entered the maximum distance desired between sections, press the **interpolate** button. When the interpolation has finished, the interpolated cross sections will automatically be drawn onto the graphic for visual inspection. An example of this is shown in Figure 6.28.

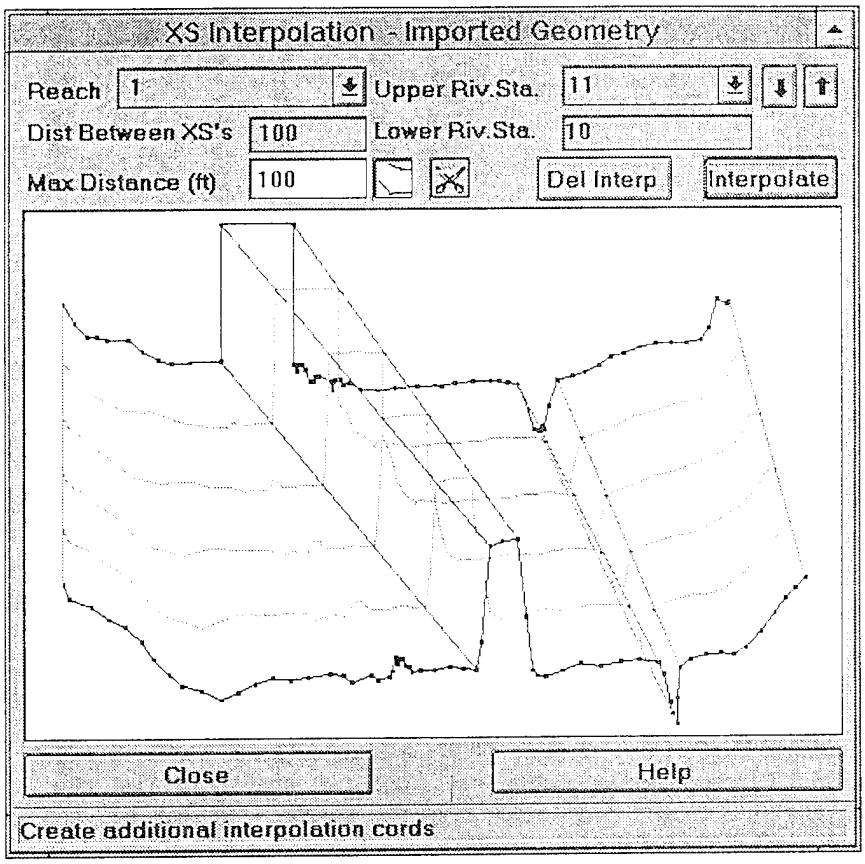


Figure 6.28 Final Interpolation With Additional Master Cords

As shown in Figure 6.28, the interpolation with the addition of user defined master cords is very reasonable.

In general, the best approach for cross section interpolation is to first interpolate sections using the "**Within a Reach**" method. This provides for fast interpolation at all locations within a reach. The "Within a Reach" method uses the five default master cords, and is usually very reasonable for most cross sections. Once this is accomplished, all of the interpolated sections should be viewed to ensure that a reasonable interpolation was accomplished in between each of the cross sections. This can be done from the "**Between 2 XS's**" window. Whenever the user finds interpolated cross sections that are not adequate, they should be deleted. A new set of interpolated sections can then be developed by adding additional master cords in order to improve the interpolation.

CAUTION: Automatic geometric cross section interpolation should not be used as a replacement for required cross section data. If water surface profile information is required at a specific location, surveyed cross section data should be provided at that location. It is very easy to use the automatic cross section interpolation to generate cross sections. But if these cross sections are not an adequate depiction of the actual geometry, you may be introducing

error into the calculation of the water surface profile. Whenever possible, use topographic maps to assist you in evaluating whether or not the interpolated cross sections are adequate. Also, once the cross sections are interpolated, they can be modified just like any other cross section.

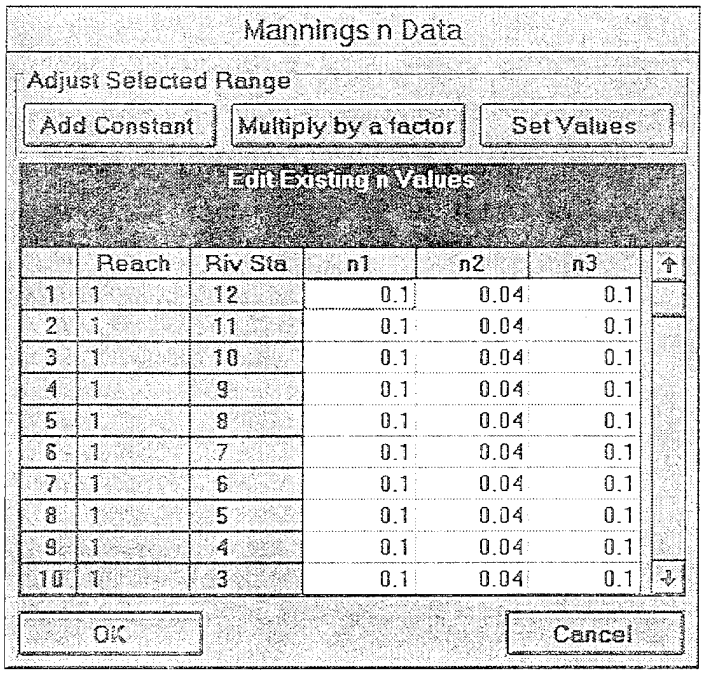
If the geometry between two surveyed cross sections does not change linearly, then the interpolated cross sections will not adequately depict what is in the field. When this occurs, the modeler should either get additional surveyed cross sections, or adjust the interpolated sections to better depict the information from the topographic map.

Viewing and Editing Data Through Tables

Once cross section data are entered, the user can view and edit certain types of data in a tabular format. The current version of HEC-RAS allows the user to view and edit Manning's n values, cross section reach lengths, and the contract and expansion coefficients in tabular form. This option is available from the **Geometric Data** editor.

Manning's n values

It is often desirable to view and edit the Manning's n values for several cross sections all at the same time. From the **Geometric Data** editor, the user can select **Manning's n values** from the **Tables** menu item. Once this option is selected, a window will appear as shown in Figure 6.29.



	Reach	Riv Sta	n1	n2	n3
1	1	12	0.1	0.04	0.1
2	1	11	0.1	0.04	0.1
3	1	10	0.1	0.04	0.1
4	1	9	0.1	0.04	0.1
5	1	8	0.1	0.04	0.1
6	1	7	0.1	0.04	0.1
7	1	6	0.1	0.04	0.1
8	1	5	0.1	0.04	0.1
9	1	4	0.1	0.04	0.1
10	1	3	0.1	0.04	0.1

Figure 6.29 Manning's n Data View and Editing Table

As shown in Figure 6.29, the user has the options to add a constant to one or more n values, multiply a group of n values by a factor, or change a group of n values to a specific value.

To add a constant to a group of n values, the user must first highlight the n values that they would like to change. Highlighting is accomplished by placing the mouse in the upper left cell of the desired cells to highlight, then press the left mouse button and drag the cursor to the lower left corner of the desired cells to highlight. When the left mouse button is released, the cells that are selected will be highlighted (except the first cell). Once the user has highlighted the desired cells to be modified, press the **Add Constant** button. This will bring up a popup window, which will allow the user to enter a constant value that will be added to all cells that are highlighted.

To multiple a group of n values by a factor, the user first highlights the desired cells. Once the cells are highlighted, pressing the **Multiply by a Factor** button will bring up a popup window. This window allows the user to enter a value that will be multiplied by each of the highlighted cells.

To set a group of n values to the same number, the user must first highlight the n values that they would like to change. Once the cells are highlighted, pressing the **Set Values** button will bring up a popup window. This window will allow the user to enter a specific n value which will replace all of the highlighted n values.

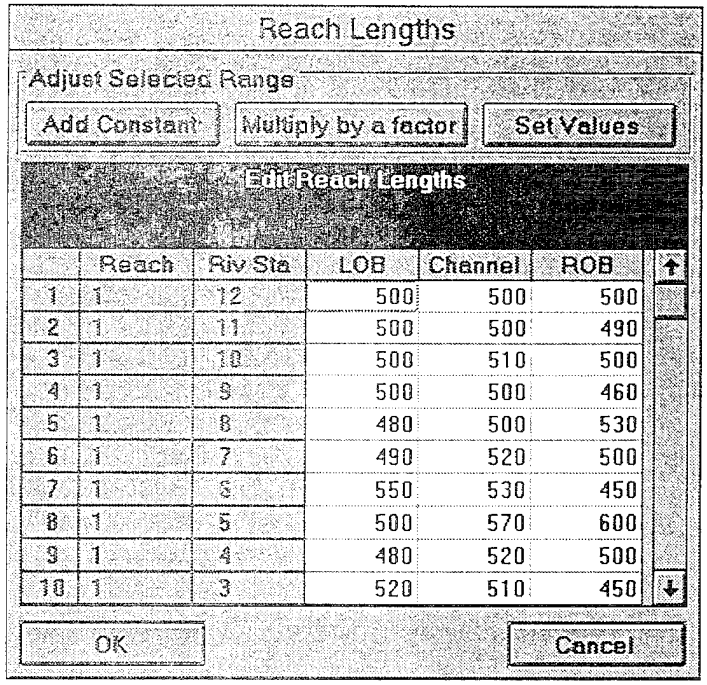
The user can also go directly into the table and change any individual n value.

Reach Lengths

The user has the ability to view and edit cross section reach lengths in a tabular format. This is accomplished by selecting **Reach Lengths** from the **Tables** menu of the Geometric Data editor. Once this option is selected, a window will appear as shown in Figure 6.30. The user has the same editing features as described previously for the n values table. See the discussion under Manning's n values, in the previous section, for details on how to edit the data.

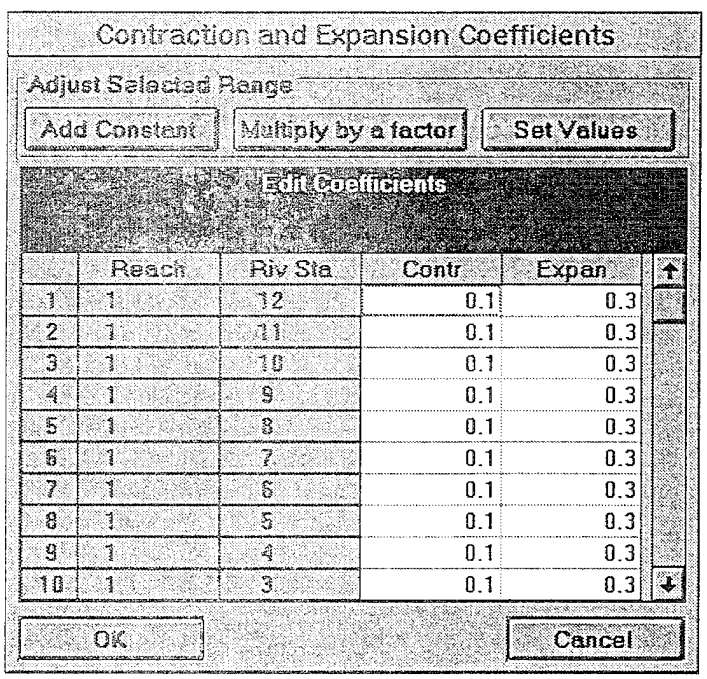
Contraction and Expansion Coefficients

The user has the ability to view and edit contraction and expansion coefficients in a tabular format. This is accomplished by selecting **Coefficients** from the **Tables** menu of the Geometric Data editor. Once this option is selected, a window will appear as shown in Figure 6.31. The user has the same editing features as described previously for the n values table. See the discussion under Manning's n values, in the previous section, for details on how to edit the data.



	Reach	Riv Sta	LOB	Channel	ROB
1	1	12	500	500	500
2	1	11	500	500	490
3	1	10	500	510	500
4	1	9	500	500	460
5	1	8	480	500	530
6	1	7	490	520	500
7	1	6	550	530	450
8	1	5	500	570	600
9	1	4	480	520	500
10	1	3	520	510	450

Figure 6.30 Reach Lengths View and Editing Table



	Reach	Riv Sta	Contr	Expan
1	1	12	0.1	0.3
2	1	11	0.1	0.3
3	1	10	0.1	0.3
4	1	9	0.1	0.3
5	1	8	0.1	0.3
6	1	7	0.1	0.3
7	1	6	0.1	0.3
8	1	5	0.1	0.3
9	1	4	0.1	0.3
10	1	3	0.1	0.3

Figure 6.31 Contraction and Expansion Coefficients Table

Saving the Geometric Data

To save the geometric data, use the **Save Geometry Data As** option from the File menu of the Geometric Data window. When this option is selected, the user is prompted to enter a title for the geometric data. Once you have entered the title, press the **OK** button and the data will be saved to the hard disk. If the geometric data have been saved before (and therefore a title has already been entered), then it is only necessary to select the **Save Geometry Data** option. When this option is selected, the geometry data are saved with the previously defined title.

In general, it is a good idea to periodically save your data as you are entering them. This will prevent the loss of large amounts of information in the event of a power failure, or if a program error occurs in the HEC-RAS user interface.

CHAPTER 7

Performing a Steady Flow Analysis

This chapter discusses how to calculate steady flow water surface profiles. The chapter is divided into two parts. The first part discusses how to enter steady flow data and boundary conditions. The second part discusses how to develop a plan and perform the calculations.

Contents

- Entering and Editing Steady Flow Data
- Performing Steady Flow Calculations

Entering and Editing Steady Flow Data

Once all of the geometric data are entered, the modeler can then enter any steady flow data that are required. To bring up the steady flow data editor, select **Steady Flow Data** from the **Edit** menu on the HEC-RAS main window. The steady flow data editor should appear as shown in Figure 7.1

Steady Flow Data

The user is required to enter the following information: the number of profiles to be calculated; the peak flow data (at least one flow for every river reach and every profile); and any required boundary conditions. The user should enter the number of profiles first. The next step is to enter the flow data. Flow data are entered directly into the table. Use the mouse pointer to select the box in which to enter the flow, then type in the desired flow value.

Flow data are entered from upstream to downstream for each reach. At least one flow value must be entered for each reach in the river system. Once a flow value is entered at the upstream end of a reach, it is assumed that the flow remains constant until another flow value is encountered within the reach. The flow data can be changed at any cross section within a reach. To add a flow change location to the table, first select the reach in which you would like to change the flow (from the reach box above the table). Next, select the River Station location for which you want to enter a flow change. Then press the **Add Flow Change Location** button. The new flow change location will appear in the table.

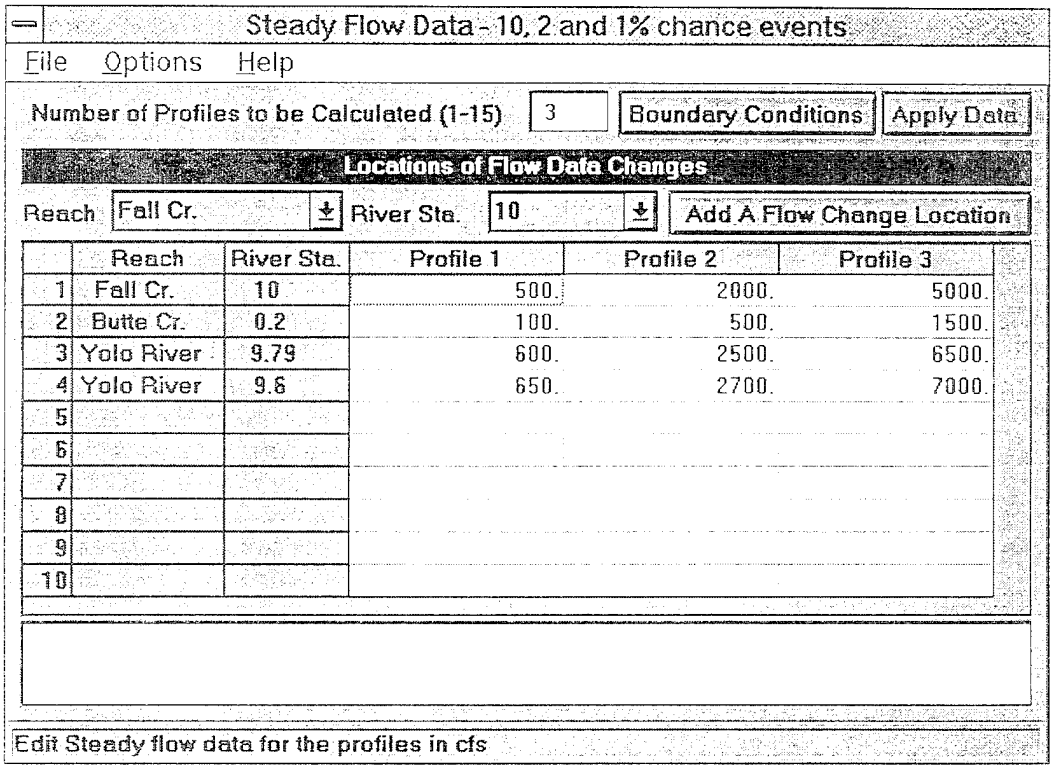


Figure 7.1 Steady Flow Data Editor

Boundary Conditions

After all of the flow data have been entered into the table, the next step is to enter any boundary conditions that may be required. To enter boundary conditions data, press the **Boundary Conditions** button at the top right of the steady flow data editor. The boundary conditions editor should appear as shown in Figure 7.2.

Boundary conditions are necessary to establish the starting water surface at the ends of the river system (upstream and downstream). A starting water surface is necessary in order for the program to begin the calculations. In a subcritical flow regime, boundary conditions are only necessary at the downstream ends of the river system. If a supercritical flow regime is going to be calculated, boundary conditions are only necessary at the upstream ends of the river system. If a mixed flow regime calculation is going to be made, then boundary conditions must be entered at all ends of the river system.

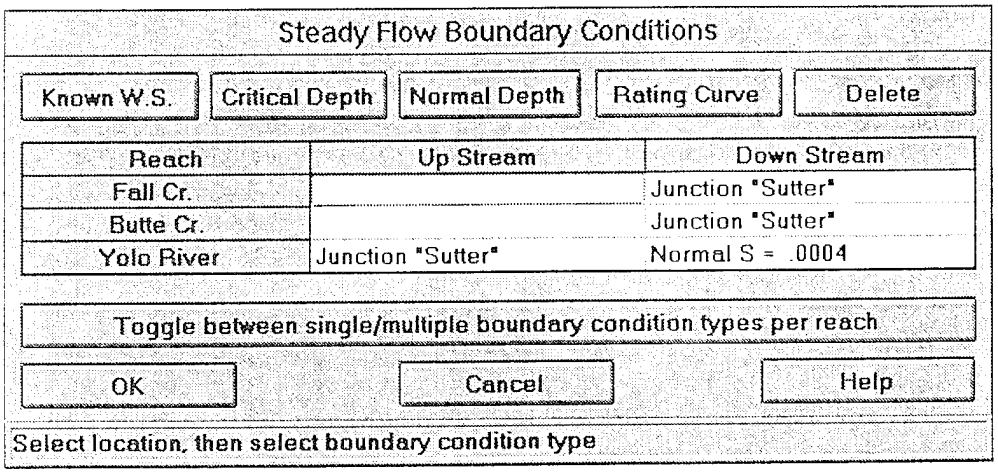


Figure 7.2 Steady Flow Boundary Conditions Editor

The boundary conditions editor contains a table listing every reach. Each reach has an upstream and a downstream boundary condition. Connections to junctions are considered internal boundary conditions. Internal boundary conditions are automatically listed in the table, based on how the river system was defined in the geometric data editor. The user is only required to enter the necessary external boundary conditions.

To enter a boundary condition, first use the mouse pointer to select the cell location in which you would like to enter a boundary condition. Then select the type of boundary condition from the four available types listed above the table. The four types of boundary conditions consist of:

Known Water Surface Elevations - For this boundary condition the user must enter a known water surface for each of the profiles to be computed.

Critical Depth - When this type of boundary condition is selected, the user is not required to enter any further information. The program will calculate critical depth for each of the profiles and use that as the boundary condition.

Normal Depth - For this type of boundary condition, the user is required to enter an energy slope that will be used in calculating normal depth (Manning's equation) at that location. A normal depth will be calculated for each profile based on the user entered slope.

Rating Curve - When this type of boundary condition is selected, a popup window appears allowing the user to enter an elevation versus flow rating curve. For each profile, the elevation is interpolated from the rating curve given the flow.

An additional feature of the boundary condition data, is that the user can specify a different type of boundary condition for each profile at the same location. This is accomplished by first pressing the button that says **Toggle between single/multiple boundary condition types per reach**. When this button is pressed, the table will expand out to provide a row for each profile, at every location. The user can then select the location and profile for which they would like to change the boundary condition type.

Once all the boundary conditions data are entered, press the **OK** button to return to the steady flow data editor. Press the **Apply Data** button to have the data accepted.

Steady Flow Data Options

Several options are available from the steady flow data editor to assist users in entering the data. These features can be found under the **Options** menu at the top of the window. The following options are available:

Undo Editing. This option allows the user to retrieve the data back to the form that it was in the last time the Apply Data button was pressed. Each time the Apply Data button is pressed, the Undo Editing feature is reset to the current information.

Delete Row From Table. This option allows the user to delete a row from the flow data table. To use this option, first select the row to be deleted with the mouse pointer. Then select **Delete Row From Grid** from the options menu. The row will be deleted and all rows below it will move up one.

Ratio Selected Flows. This option allows the user to multiply selected values in the table by a factor. Using the mouse pointer, hold down the left mouse button and highlight the cells that you would like to change by a factor. Next, select **Ratio Selected Flows** from the options menu. A popup window will appear allowing you to enter a factor to multiply the flows by. Once you press the **OK** button, the highlighted cells will be updated with the new values.

Specify Change in Energy. This option allows the user to force a specific change in energy between any to cross sections in the model. The change in energy can be set for a specific profile in a multiple profile model. As shown in Figure 7.3, the user first selects a the river reach. Then the from river station (from RS) and to river station (to RS) within the reach for which the energy loss will be applied. Next, the user selects the profile number and enters the change in energy. The final step is to press the "set" button to add the change in energy to the table. The user can set several changes in energy within the model, for various profiles. During the computations, the program will use the specifyed change in energy between two section, and then compute a water surface for that energy at the upstream section.

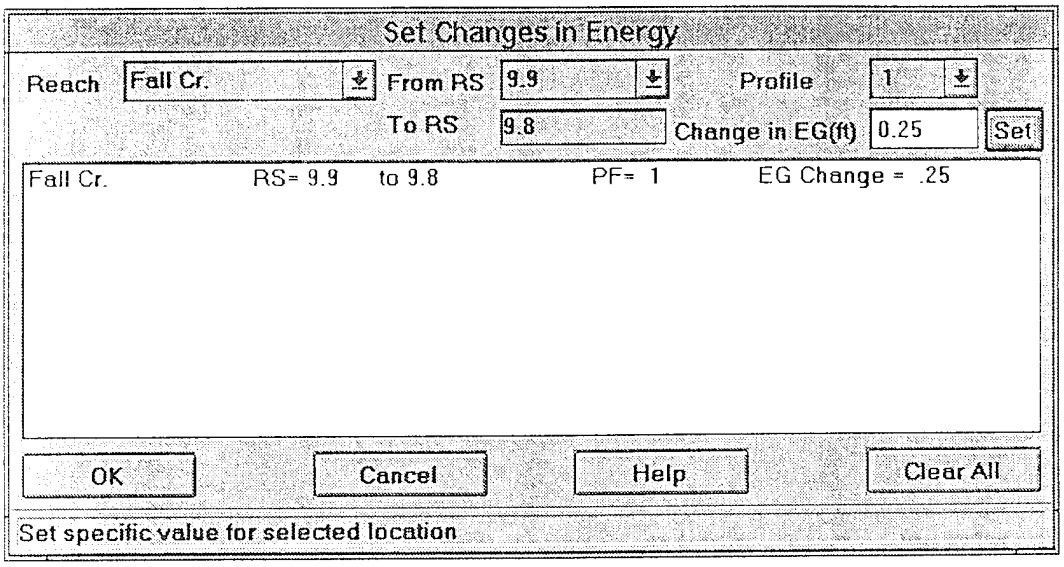


Figure 7.3 Setting a Change in Energy Between Two Cross Sections

Specify Change in WS. This option allows the user to force a specific change in the water surface elevation between two cross sections. This option works exactly the same as the "Specify Change in Energy" option described previously. The user can specify changes in water surface elevations at multiple locations for various profiles.

Inset Known WS. This option allows the user to force a specific water surface at a given cross section. The known water surface elevations can be specified at multiple locations for various profiles. During the computations, the program will not compute a water surface elevation for any cross section where a known water surface elevation has been entered. The program will use the known water surface elevation and then move to the next section.

Additional EG Losses. This option allows the user to enter an additional energy loss between two cross sections. This energy loss gets added to any computed energy losses that occur during the balancing of the energy equation. The additional energy loss can be specified at multiple locations for various profiles.

Saving The Steady Flow Data

The last step in developing the steady flow data is to save the information to a file. To save the data, select the **Save Flow Data As** from the **File** menu on the steady flow data editor. A popup window will appear prompting you to enter a title for the data.

Performing Steady Flow Calculations

Once all of the geometry and steady flow data have been entered, the user can begin calculating the steady flow water surface profiles. To perform the simulations, go to the HEC-RAS main window and select Steady Flow **Analysis** from the **Simulate** menu. The Steady Flow Analysis window should appear as in Figure 7.4.

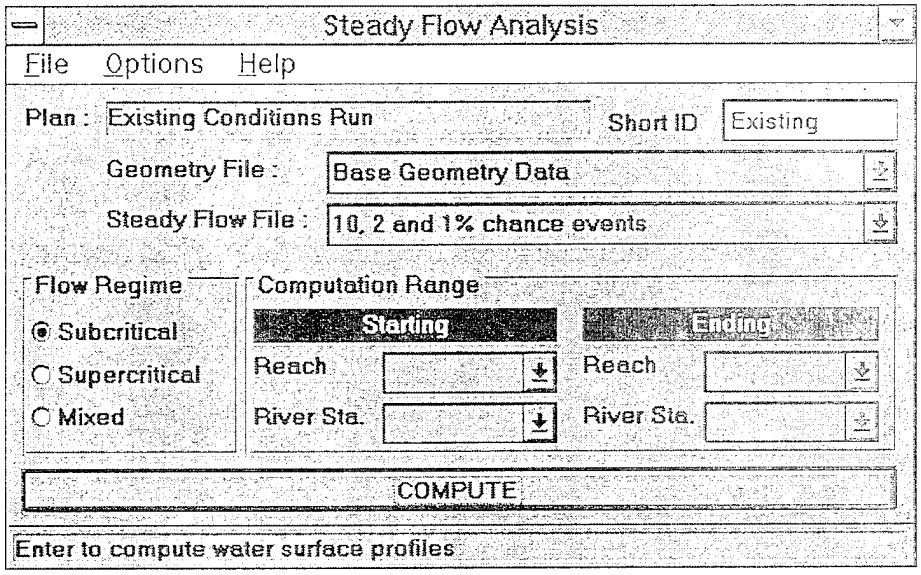


Figure 7.4 Steady Flow Analysis window

Defining A Plan

The first step in performing a simulation is to put together a Plan. The Plan defines which geometry and flow data are to be used, as well as provide a description and short identifier for the run. Also included in the plan information are the selected flow regime and the simulation options.

To establish a Plan, select **New Plan** from the **File** menu on the steady flow analysis window. When **New Plan** is selected, a window will appear prompting you to enter a title for the plan. After you enter the title, press the **OK** button to close the window and accept the title. The user can also enter a short identifier for the plan. This identifier is optional. To enter the short identifier, simply click the left mouse button on the identifier box and enter the name. The short identifier is limited to 12 characters.

Once the title and short identifier are established, the next step is specify geometry and flow data to be used in the plan. To select a geometry or flow file, press the down arrow button next to the desired data type. When this

button is pressed, a list will appear displaying all of the available files of that type that are currently available for the project. Select the geometry and flow files that you want to use for the current plan.

The last step is to select the desired flow regime for which the model will perform calculations. The user can select between subcritical, supercritical, or mixed flow regime calculations.

Saving The Plan Information

To save the Plan information, select **Save Plan** from the **File** menu of the simulation window.

Simulation Options

The following is a list of the available simulation options under the **Options** menu of the steady flow simulations window:

Encroachments. This option allows the user to perform a floodway encroachment analysis. For a detailed description of how to use the floodway encroachment capabilities of HEC-RAS, see Chapter 9 of the User's Manual (this manual). For a description of how the encroachment calculations are performed for the various encroachment methods, see Chapter 8 of the Hydraulic Reference Manual.

Flow Distribution Locations. This option allows the user to specify locations in which they would like the program to calculate flow distribution output. The flow distribution option allows the user to subdivide the left overbank, main channel, and right overbank, for the purpose of computing additional hydraulic information.

The flow distribution output can be obtained by first defining the locations that the user would like to have this type of output. As shown in Figure 7.5, the user can either select specific locations or all locations in the model. Next, the number of slices for the flow distribution computations must be defined for the left overbank, main channel, and the right overbank. The user can define up to 45 total slices. Each flow element (left overbank, main channel, and right overbank) must have at least one slice. The user can change the number of slices used at each of the cross sections. The flow distribution output will be calculated for all profiles in the plan during the computations.

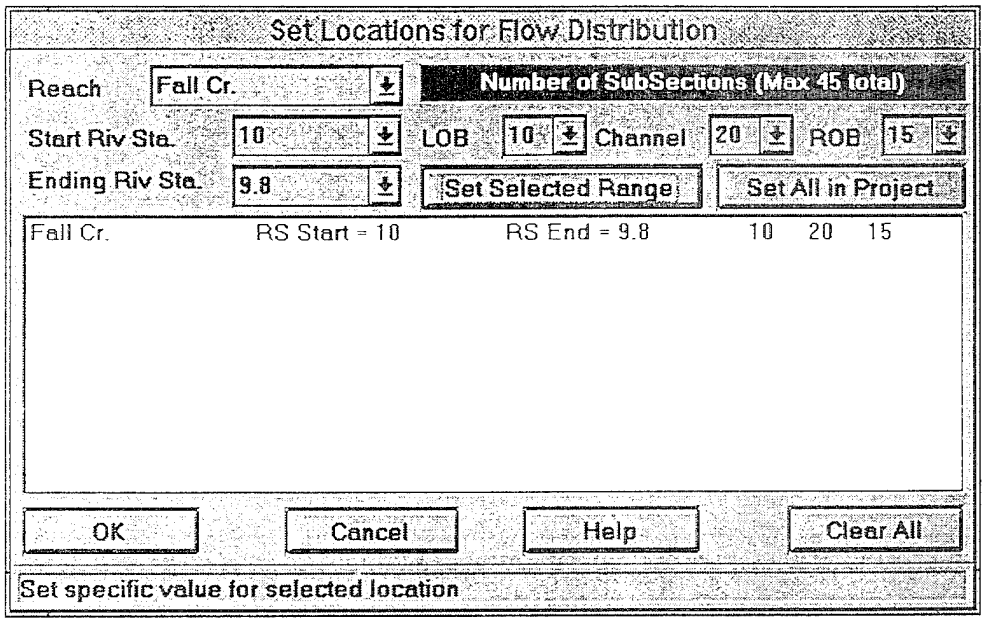


Figure 7.5 Window for Specifying the Locations of Flow Distribution.

During the normal profile computations, at each cross section where flow distribution is requested, the program will calculate the percentage of flow, area, wetted perimeter, conveyance, and average velocity for each of the user defined slices. For details on how the flow distribution output is actually calculated, see Chapter 4 of the HEC-RAS Hydraulic Reference Manual. For information on viewing the flow distribution output, see Chapter 8 of the User's Manual (this manual).

Set Output Options. This option allows the user to request some additional output that is not normally computed. Currently there is only one choice available, which is to have the program calculate critical depth at all locations.

Conveyance Calculations. This option allows the user to tell the program how to calculate conveyance in the overbanks. Two options are available. The first option, **At breaks in n values only**, instructs the program to sum wetted perimeter and area between breaks in n values, and then to calculate conveyance at these locations. If n varies in the overbank the conveyance values are then summed to get the total overbank conveyance. The second option, **Between every coordinate point (HEC2 style)**, calculates wetted perimeter, area, and conveyance between every coordinate point in the overbanks. The conveyance values are then summed to get the total left overbank and right overbank conveyance. These two methods can provide different answers for conveyance, and therefore different computed water surfaces. The **At breaks in n values only** method is the default.

Friction Slope Methods. This option allows the user to select one of four available friction slope equations, or to allow the program to select the method based on the flow regime and profile type. The four equations are:

- Average Conveyance (Default)
- Average Friction Slope
- Geometric Mean Friction Slope
- Harmonic Mean Friction Slope

Set Calculation Tolerances. This option allows the user to override the default settings for the calculation tolerances. These tolerances are used in the solution of the energy equation. **Warning !!!** - Increasing the default calculation tolerances could result in computational errors in the water surface profile. The tolerances are as follows:

Water surface calculation tolerance: This tolerance is used to compare against the difference between the computed and assumed water surface elevations. When the difference is less than the tolerance, the program assumes that it has a valid numerical solution.

Critical depth calculation tolerance: This tolerance is used during the critical depth solution algorithm.

Maximum number of iterations: This variable defines the maximum number of guesses that the program will make when attempting to balance a water surface.

Maximum difference tolerance: This tolerance is used during the balance of the energy equation. As the program attempts to balance the energy equation, the solution with the minimum error (assumed minus computed water surface) is saved. If the program goes to the maximum number of iterations without meeting the specified calculation tolerance, the minimum error solution is checked against the maximum difference tolerance. If the solution at minimum error is less than this value, then the program uses the minimum error solution as the answer, issues a warning statement, and then proceeds with the calculations. If the solution at minimum error is greater than the maximum difference tolerance, then the program issues a warning and defaults the solution to critical depth. The computations then proceed from there.

Flow Tolerance Factor: This factor is only used in the bridge and culvert routines. The factor is used when the program is attempting to balance between weir flow and flow through the structure. The factor is multiplied by the total flow. The resultant is then used as a flow tolerance for the balance of weir flow and flow through the structure.

Each of these variables has an allowable range and a default value. The user is not allowed to enter a value outside of the allowable range.

Critical Depth Computation Method. This option allows the user to select between two methods that can be used for calculating critical depth during the computations. The default method is the **Parabolic Method**. This method utilizes a parabolic searching technique to find the minimum specific energy. This method is very fast, but it is only capable of finding a single minimum on the energy curve. A second method, **Multiple Critical Depth Search**, is capable of finding up to three minimums on the energy curve. If more than one minimum is found the program selects the answer with the lowest energy. Very often the program will find minimum energies at levee breaks and breaks due to ineffective flow settings. When this occurs, the program will not select these answers as valid critical depth solutions, unless there is no other answer available. The Multiple Critical Depth Search routine takes a lot of computation time. Since critical depth is calculated often, using this method will slow down the computations. This method should only be used when you feel the program is finding an incorrect answer for critical depth.

Check Data Before Execution. This option provides for comprehensive data input checking. When this option is turned on, data input checking will be performed when the user presses the compute button. If all of the data are complete, then the program allows the steady flow computations to proceed. If the data are not complete, or some other problem is detected, the program will not perform the steady flow analysis, and a list of all the problems in the data will be displayed on the screen. If this option is turned off, data checking is not performed before the steady flow execution. The default is that the data checking is turned on.

Set Log File Output Level. This option allows the user to set the level of the Log file. The Log file is a file that is created by the computational program. This file contains information tracing the program process. Log levels can range between 0 and 10, with 0 resulting in no Log output and 10 resulting in the maximum Log output. In general, the Log file output level should not be set unless the user gets an error during the computations. If an error occurs in the computations, set the log file level to an appropriate value. Re-run the computations and then review the log output, try to determine why the program got an error.

When the user selects **Set Log File Output Level**, a window will appear as shown in Figure 7.6. The user can set a "Global Log Level", which will be used for all cross sections and every profile. The user can also set log levels at specific locations for specific profiles. In general, it is better to only set the log level at the locations where problems are occurring in the computations. To set the specific location log level, first select the desired reach and river station. Next select the log level and the profile number (the

log level can be turned on for all profiles). Once you have everything set, press the **Set** button and the log level will show up in the window below. Log levels can be set at several locations individually. Once all of the Log Levels are set, press the **OK** button to close the window.

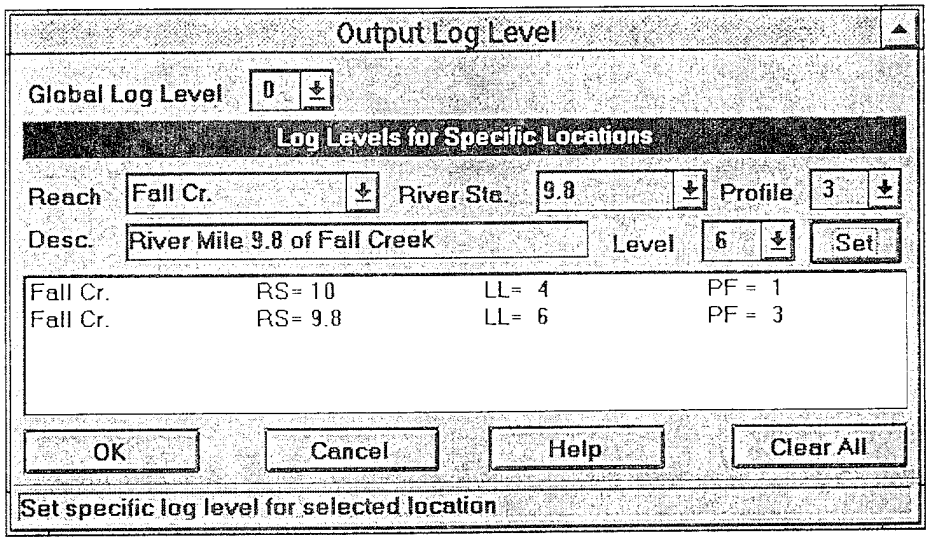


Figure 7.6 Log File Output Level window

Warning !!! - setting the log output level to 4 or 5 can result in very large log file output. Log level values of 6 or larger can result in extremely large log files.

View Log File. This option allows the user to view the contents of the log file. The interface uses the Windows Write program to accomplish this. It is up to the user to set an appropriate font in the Write program. If the user sets a font that uses proportional spacing, the information in the log file will not line up correctly. Some fonts that work well are: Line Printer; Courier (8 pt.); and Helvetica (8 pt.). Consult your Windows user's manual for information on how to use the Write program.

Starting the Computations

Once all of the data have been entered, and a Plan has been defined, the steady flow computations can be performed by pressing the **Compute** button at the bottom of the steady flow simulation window. Once the compute button is pressed, a separate window will appear showing you the progress of the computations. The information that appears in the window is only there as an indicator of the programs progress during the computations. This window is not intended to be used for viewing any output. When the computations have been completed, the user can close the computations window by double clicking the upper left corner of the window. If the

computations ended with a message stating "**PROGRAM TERMINATED NORMALLY**", the user can then begin to review the output.

CHAPTER 8

Viewing Results

After the model has finished the steady flow profile computations, you can begin to view the output. Output is available in a graphical and tabular format. The current version of the program allows the user to view cross sections, profiles, rating curves, X-Y-Z perspective plots, detailed tabular output at a single cross section, and limited tabular output at many cross sections. Users also have the ability to develop their own output tables.

Contents

- Cross Sections, Profiles, and Rating Curves
- X-Y-Z Perspective Plots
- Tabular Output
- View Results from the River System Schematic
- Viewing Flow Distribution Output

Cross Sections, Profiles, and Rating Curves

Graphical displays are often the most effective method of presenting input data and computed results. Graphics allow the user to easily spot errors in the input data, as well as providing an overview of the results in a way that tables of numbers cannot.

Viewing Graphics on the Screen

To view a graphic on the screen, select either **Cross Sections, Profiles,** or **Rating Curves** from the **View** menu on the HEC-RAS main window. Once you have selected one of these options, a window will appear with the graphic plotted in the viewing area. An example cross section plot is shown in Figure 8.1. The user can plot any cross section by simply selecting the appropriate reach and river station from the list boxes at the top of the plot. The user can also step through the cross section plots by using the up and down arrow buttons next to the river station box.

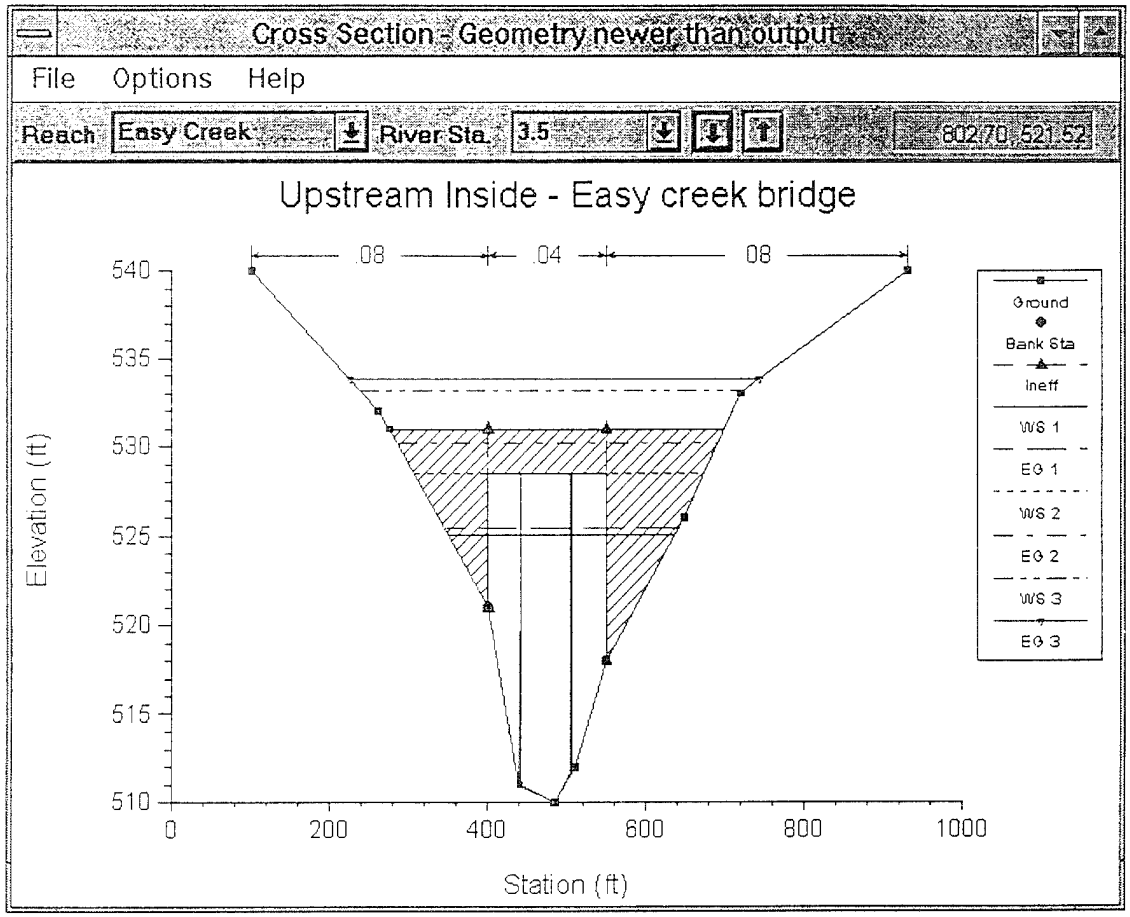


Figure 8.1 Example Cross Section Plot

An example profile plot is shown in Figure 8.2. The profile plot displays the water surface profile for the first reach in the river system. If there is more than one reach, additional reaches can be selected from the Options menu on the plot.

An example rating curve plot is shown in Figure 8.3. The rating curve is a plot of the water surface elevation versus flow rate for the profiles that were computed. A rating curve can be plotted at any location by selecting the appropriate reach and river station from the list boxes at the top of the plot.

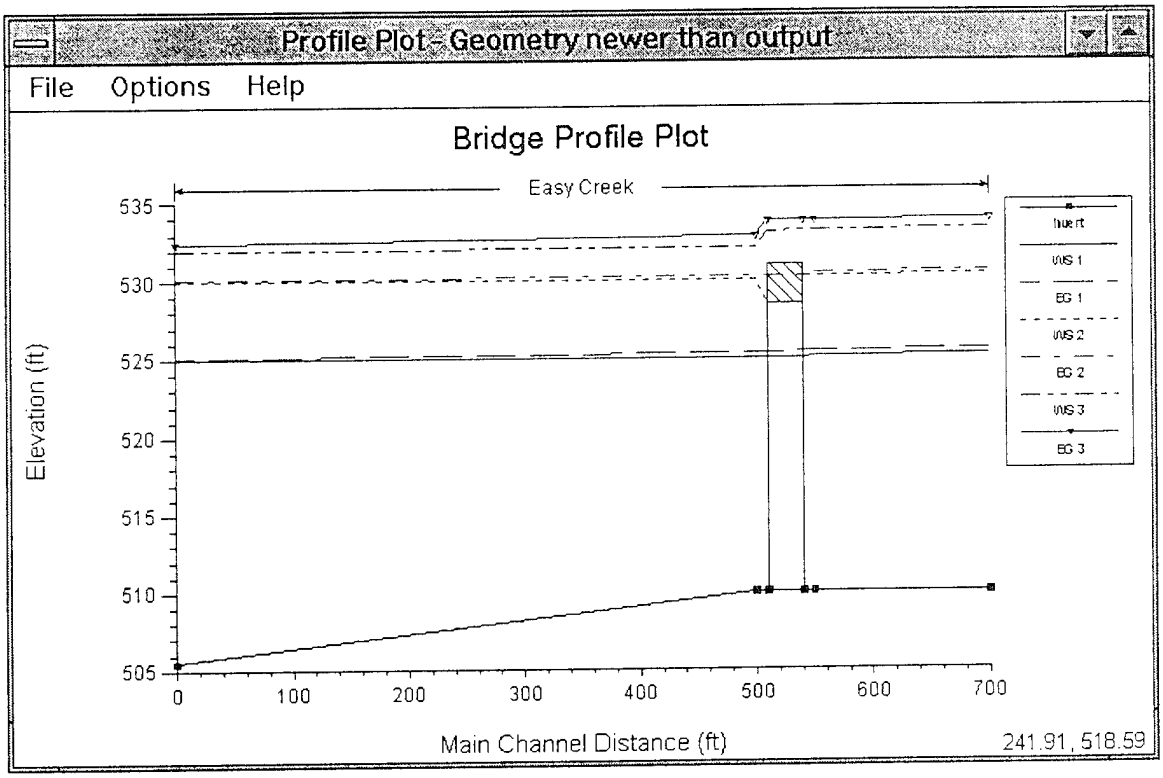


Figure 8.2 Example Profile Plot

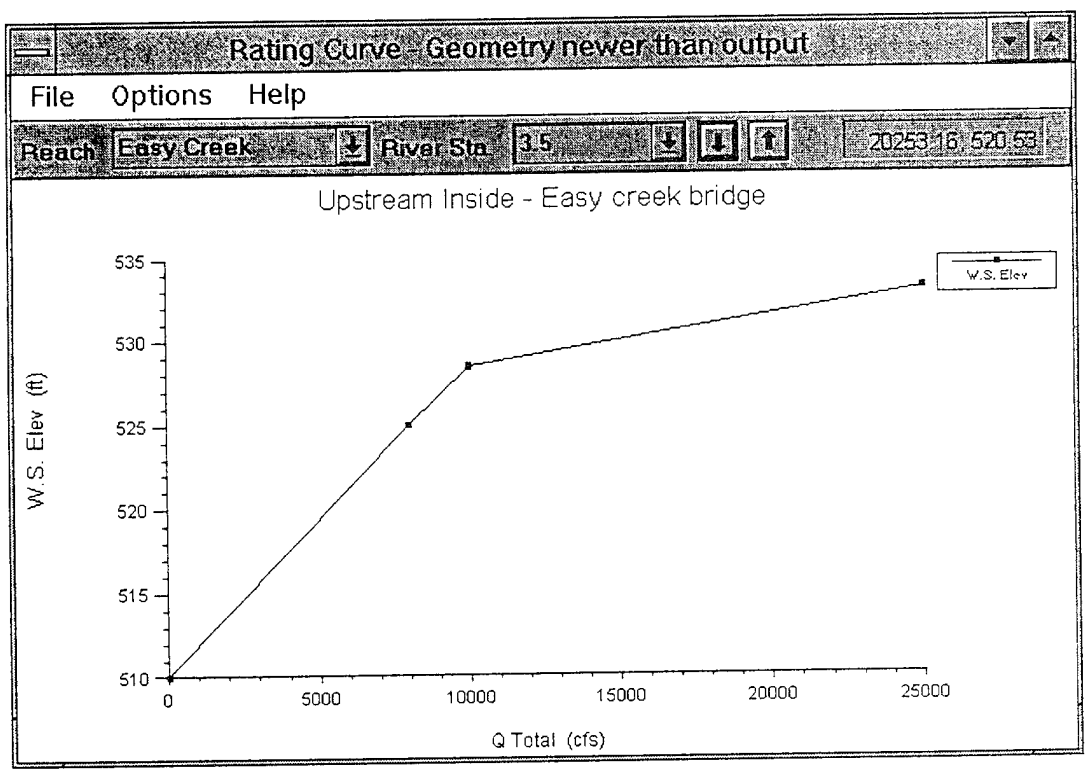


Figure 8.3 Example Rating Curve Plot

Graphical Plot Options

Several plotting features are available from the **Options** menu on all of the graphical plots. These options include: zoom in; zoom out; selecting which plans, profiles, reaches and variables to plot; and control over labels, lines, symbols, scaling, and grid options. In general, the options are about the same on all of the graphics.

Zoom In: This option allows the user to zoom in on a portion of the graphic. This is accomplished by selecting **Zoom In** from the **Options** menu, then specifying the area to zoom in on with the mouse. Defining the zoom area is accomplished by placing the mouse pointer at a corner of the desired zoom area. Then press down on the left mouse button and drag the mouse to define a box containing the desired zoom area. Finally, release the left mouse button and the viewing area will display the zoomed-in area of the graphic.

Zoom Out: This option re-displays the graphic back into its original size before you zoomed in. Zooming out is accomplished by selecting **Zoom Out** from the **Options** menu.

Plans. This option allows the user to select from the available Plans for plotting. The default plan is the currently opened plan. The user can select additional plans to view for comparison of results graphically.

Profiles. This option allows the user to select which profiles they would like to have displayed on the graphic. This option does not apply to the rating curve, it automatically plots all of the profiles.

Reaches. This option allows the user to select which river reaches they would like to have displayed. This option only applies to the profile plot.

Variables. This option allows the user to select what ever variables are available for plotting. The number and type of variables depends on what type of graphic is being displayed.

Labels. This option allows the user to change the labels on the plot. This includes the plot caption, the X axis title, and the Y axis title.

Legend. This option allows the user to change the labels of the legend.

Lines and Symbols. This option allows the user to change the line types, line colors, line widths, symbol types, symbol colors, symbol sizes, fill patterns, and the line labels.

Scaling. This option allows the user to define the scaling used for the plot. Users are allowed to set the minimum, maximum, and labeling increment for the X and Y axis. Scaling can be set temporarily, or scaling can be set to be persistent (scaling stays constant for all cross sections). Persistent scaling is

only available for the cross section and rating curve plots.

Grid. This option allows the user to overlay a grid on top of the graphic. Users have the option to have both major and minor tics displayed.

Plotting Velocity Distribtution Output

The user has the option of plotting velocity distribution output from the cross section viewer. Velocity distributions can only be plotted at locations in which the user specified that flow distribution output be calculated during the computations. To view the velocity distribution plot, first bring up a cross section plot (select "Cross Sections" from the view menu of the main HEC-RAS window). Next, select the cross section in which you would like to see the velicty distribution output. Select **Velocity Distribution** form the **Options** menu of the cross section window. This will bring up a popup window (Figure 8.4) that will allow you to set the minimum velocity, maximum; velocity, and velocity increment for plotting. In general, it is better to let the program use the maximum velocity range for plotting. Next, the user selects **Plot Velocity Distribution**. Then press the "OK" button and the velocity distribution plot will appear as shown in Figure 8.5.

For detailes on how to select the locations for computing the velocity distribution, see Chapter 7 of the User's Manual. For information on how the velocity distribution is actually calculated, see Chapter 4 of the Hydraulic Reference Manual.

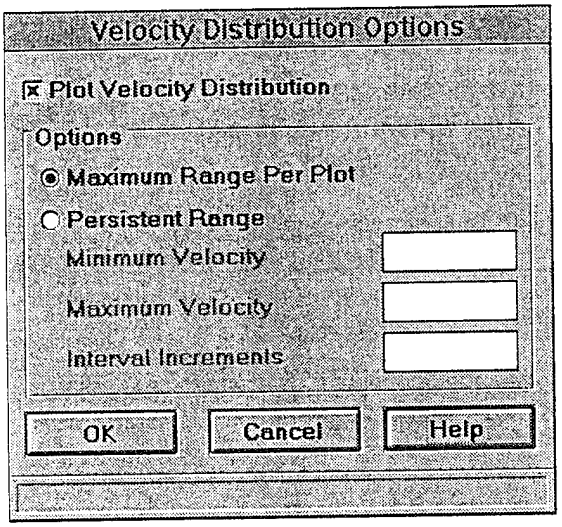


Figure 8.4 Velocity Distribution Options.

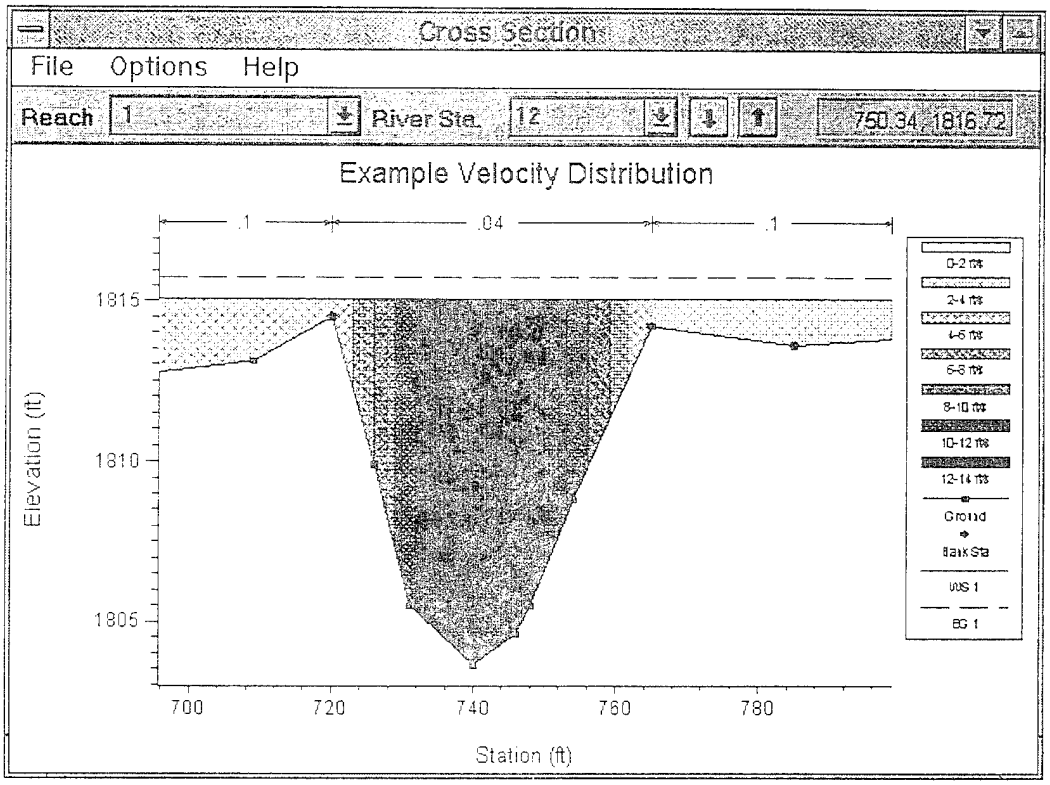


Figure 8.5 Velocity Distribution Plot

Plotting Other Variables in Profile

The profile plotting window has the ability to plot other variables, besides water surface and energy, in profile. Any variable that is computed at a cross section can be displayed in profile. An example would be to plot velocity versus distance. To plot other variables in profile, the user must first select the **General Profile** option from the **Options** menu of the Profile plot. Once this option is turned on, the plot automatically switches to a plot of channel velocity versus distance. Other variables can be selected from the **Variables** option under the **Options** menu. The user can plot several different variable types at one time (e.g. velocity and area versus distance), but the scaling may not be appropriate when this is done. An example of plotting other variables in profile is shown in Figure 8.6.

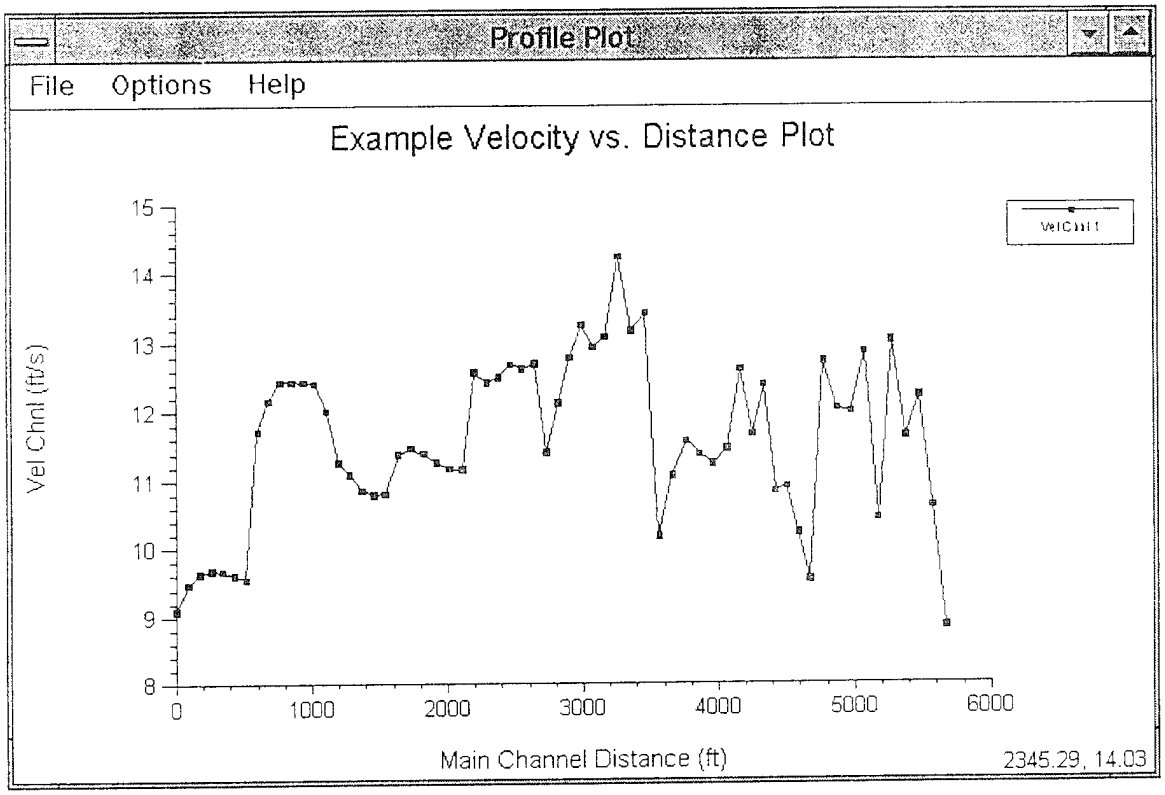


Figure 8.6 General Profile Plot of Variables Versus Distance

Plotting One Variable Versus Another

The rating curve plotting window has the ability to plot other variables besides discharge versus water surface elevation. Any variable that is computed at a cross section can be dispayed against onther computed variable (or variables). An example of this capability is shown in Figure 8.7. In this example, Discharge (x-axis) is being plotted against total flow area and main channel flow area (y-axis).

To plot other variables, the user selects the **X Axis Variable** and **Y Axis Variable** from the **Options** menu ofthe rating curve plotting window. When selecting variables to plot, keep in mind that all variables selected for a particular axis should have a similar range in magnitude.

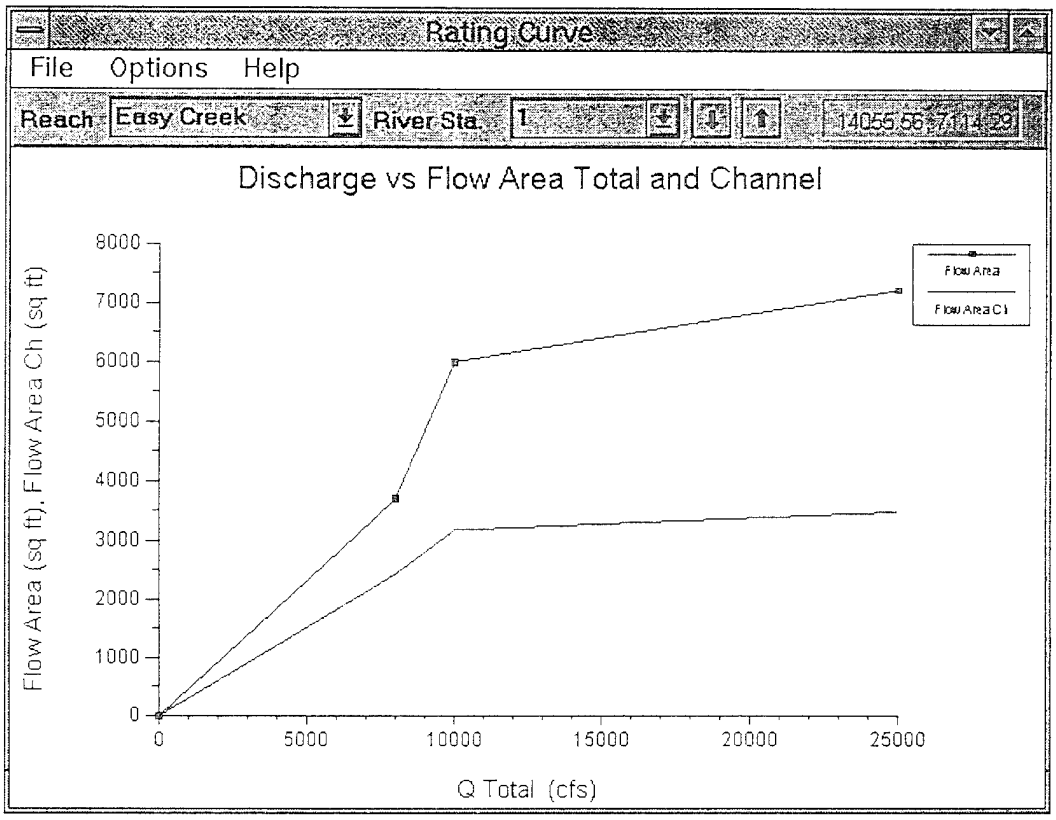


Figure 8.7 Example of Plotting One Variable Against Other Variables

Sending Graphics to the Printer or Plotter

All of the graphical plots in HEC-RAS can be sent directly to a printer or plotter. The printer or plotter used depends on what you currently have set as the default printer or plotter in the Windows Print Manager. To send a graphic to the printer or plotter, do the following:

1. Display the graphic of interest (cross section, profile, rating curve, or river system schematic) onto the screen.

2. Using the available graphics options (scaling, labels, grid, etc...), modify the plot to be exactly what you would like printed.

3. Select **Print Current** (or just **Print** on the from the profile plot) from the **File** menu of the displayed graphic. When this option is selected, a popup window will appear allowing you to modify the default print options. Change any desired options and press the **Print** button. The graphic will be sent to the Windows Print Manager. The print manager will then send the plot to the default printer or plotter.

Note: The user can print multiple cross sections at one time by using the **Print Multiple** option from the **File** Menu of the cross section and rating curve plots. This option also allows the user to establish how many cross sections or rating curves they would like to have printed on each page.

Sending Graphics to the Windows Clipboard

All of the HEC-RAS graphics can be sent to the Windows Clipboard. Passing a graphic to the clipboard allows that graphic to then be pasted into another piece of software (i.e., a word processor). To pass a graphic to the windows clipboard, and then to another program, do the following:

1. Display the graphic of interest on the screen.

2. Using the options, modify the plot to be exactly what you want.

3. Select **Copy to Clipboard** from the **File** menu of the displayed graphic. The plot will automatically be sent to the Windows Clipboard.

4. Bring up the program that you want to pass the graphic into. Select **Paste** from the **Edit** menu of the receiving program. Once the graphic is pasted in, it can be re-sized to the desired dimensions.

HEC-RAS sends and displays all graphics in a Window's Meta file format. Since Meta files are vector based graphics, the graphic can be resized without causing the image to distort.

X-Y-Z Perspective Plots

Another type of graphic available to the user is the X-Y-Z Perspective Plot. The X-Y-Z plot is a pseudo 3-dimensional plot of multiple cross sections within a reach. An example X-Y-Z Perspective plot is shown in Figure 8.8.

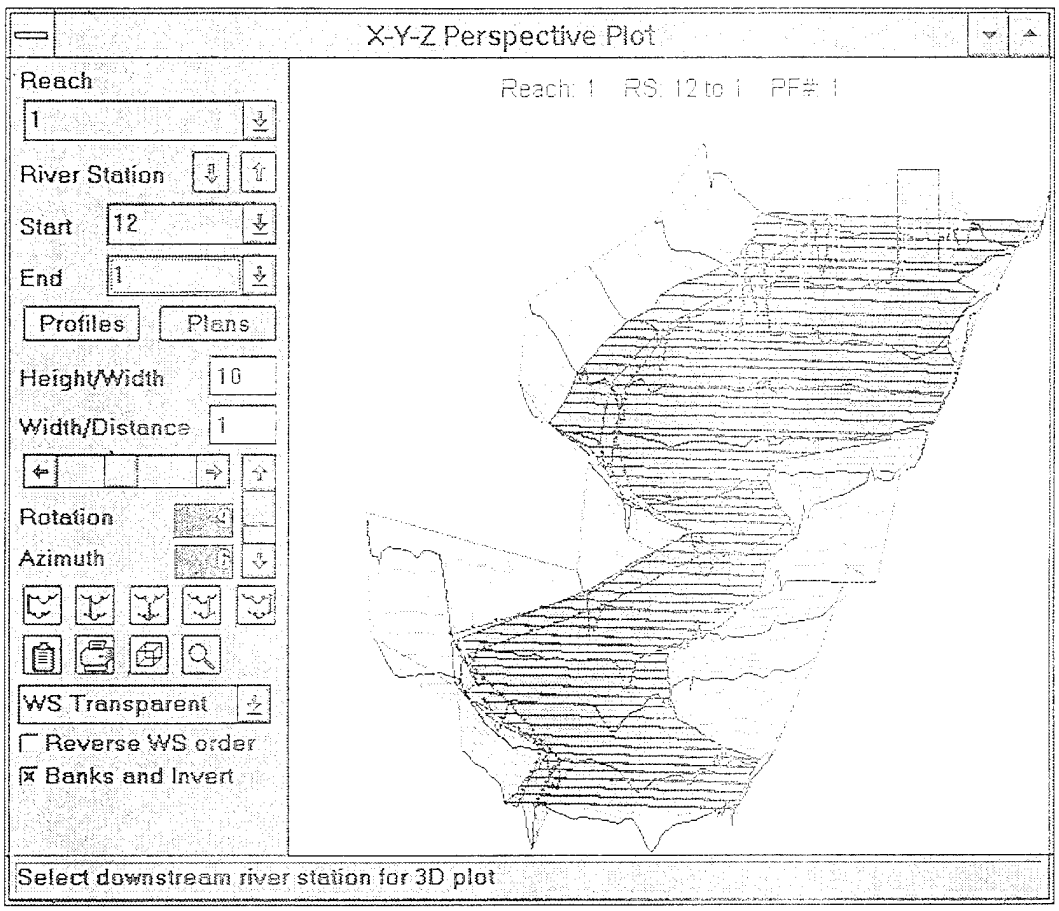


Figure 8.8 Example X-Y-Z Perspective Plot

The user has the ability to select the reach to be plotted, the range of the river stations, and which plans and profiles to be displayed. The plot can be rotated left and right, as well as up and down, in order to get different perspectives of the river system. The user can choose to overlay the water surface or not. The water surface can be displayed in an outline format, a shaded but transparent format, or a solid filled format. The graphic can be sent to the printer/plotter or the clipboard just like any other plot. Sending the graphic to the printer or clipboard is accomplished by pressing the printer or clipboard button, located on the lower left hand side of the plot. The data in the plot can

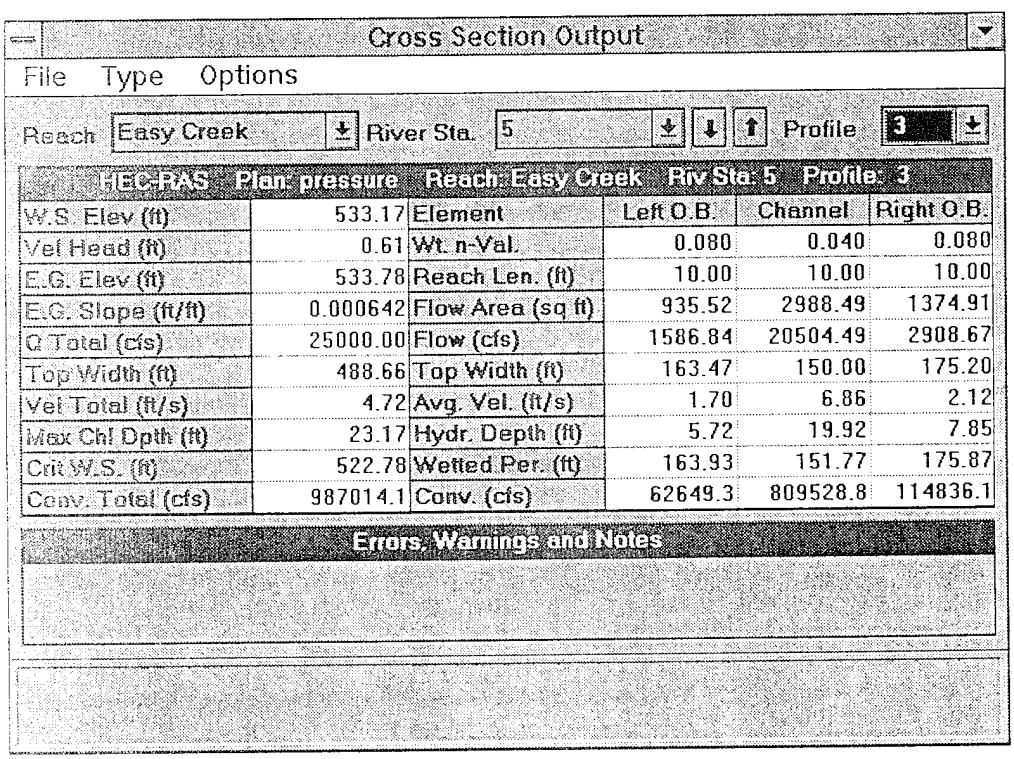
also be aligned in several different ways, including: the left edge of the cross section (default); the left bank station of the main channel; the channel inverts; the right main channel bank station; and the right edge of the cross section data. This is accomplished by using the data alignment buttons above the printer and clipboard buttons.

Tabular Output

Summary tables of the detailed water surface profile computations are often necessary to analyze and document simulation results. Tabular output allows the user to display large amounts of detailed information in a concise format. HEC-RAS has two basic types of tabular output, cross section tables and profile tables.

Cross Section Tables

Cross section tables show detailed hydraulic information at a single location, for a single profile. To display a cross section table on the screen, select **Cross Section Table** from the **View** menu of the main HEC-RAS window. An example cross section table is shown in Figure 8.9.



W.S. Elev (ft)	533.17	Element	Left O.B.	Channel	Right O.B.
Vel Head (ft)	0.61	Wt. n-Val.	0.080	0.040	0.080
E.G. Elev (ft)	533.78	Reach Len. (ft)	10.00	10.00	10.00
E.G. Slope (ft/ft)	0.000642	Flow Area (sq ft)	935.52	2988.49	1374.91
Q Total (cfs)	25000.00	Flow (cfs)	1586.84	20504.49	2908.67
Top Width (ft)	488.66	Top Width (ft)	163.47	150.00	175.20
Vel Total (ft/s)	4.72	Avg. Vel. (ft/s)	1.70	6.86	2.12
Max Ch! Dpth (ft)	23.17	Hydr. Depth (ft)	5.72	19.92	7.85
Crit W.S. (ft)	522.78	Wetted Per. (ft)	163.93	151.77	175.87
Conv. Total (cfs)	987014.1	Conv. (cfs)	62649.3	809528.8	114836.1

Figure 8.9 **Example Cross Section Table**

Any cross section can be displayed in the table by selecting the appropriate reach and river station from the list boxes at the top of the table. Also, any of the computed profiles can be displayed by selecting the desired profile from the profile list box.

Users can also view detailed hydraulic information for other types of cross section specific tables. Other table types are selected from the **Type** menu on the cross section table window. The following types are available in addition to the normal cross section table (which is the default):

Culvert. The culvert table type brings up detailed culvert information. This table can be selected for normal culverts, or for culverts that are part of a multiple opening river crossing. An example culvert specific table is shown in Figure 8.10.

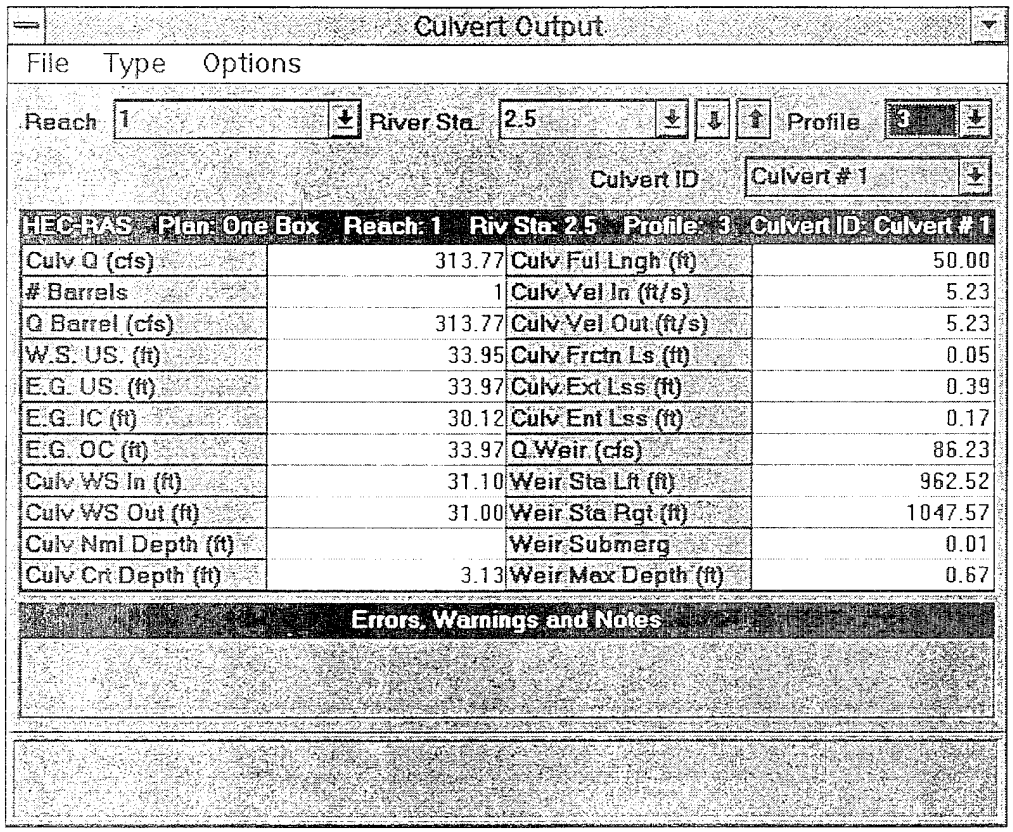


Figure 8.10 Example Culvert Type of Cross Section Table

Bridge. The bridge table type brings up detailed output for the cross sections inside the bridge as well as just upstream of the bridge. The bridge table type can selected for normal bridge crossings, or for bridges that are part of a multiple opening river crossing. An example of the bridge specific cross section table is shown in Figure 8.11.

Bridge Output

File Type Options

Reach Easy Creek River Sta. 3.5 Profile 1

Opening Bridge #1

HEC-RAS Plan: pressure		Reach: Easy Creek	Riv Sta: 3.5	Profile: 1 Opening:
E.G. US. (ft)	525.41	Element	Inside BR US	Inside BR DS
W.S. US. (ft)	525.09	E.G. Elev (ft)	525.40	525.37
Q Total (cfs)	8000.00	W.S. Elev (ft)	525.06	525.03
Q Bridge (cfs)	8000.00	Crit W.S. (ft)	517.49	517.49
Q Weir (cfs)		Max Chl Dpth (ft)	15.06	15.03
Weir Sta Lft (ft)		Vel Total (ft/s)	4.66	4.68
Weir Sta Rgt (ft)		Flow Area (sq ft)	1715.44	1710.78
Weir Submerg		Froude # Chl	0.24	0.24
Weir Max Depth		Specif Force	11908.15	11856.62
Min Top Rd (ft)	531.00	Hydr Depth (ft)	11.75	11.72

Errors, Warnings and Notes

Figure 8.11 Example Bridge Type of Cross Section Table

Conveyance. The conveyance type of table is the same as a normal cross section table. The only difference is that a conveyance type of table is used for view the hydraulic results of a portion of a multiple opening river crossing. That is, if the user has defined an open channel flow type of opening (i.e. conveyance area), at a multiple opening crossing, then this table can be used to view the hydraulic results for that specific opening.

Flow Distribution. The Flow Distribution table type can be used to view the computed flow distribution output at any cross section where this type of output was requested. An example of the flow distribution table output is shown in Figure 8.12.

Flow Distribution Output

File Type Options

Reach 1 River Sta. 8 Profile 1

HEC-RAS Plan: Existing Reach: 1 River Sta: 8 Profile: 1

Left Sta (ft)	Right Sta (ft)	%Q	Area (sq ft)	W.P. (ft)	Conv. (cfs)	Hydr D. (ft)	Velocity (ft/s)
217.80	435.60	1.67	121.17	186.34	1351.4	0.65	1.07
435.60	653.40	7.57	320.05	218.23	6138.6	1.47	1.85
653.40	871.20	16.74	514.70	217.84	13567.6	2.36	2.54
871.20	1089.00	12.95	441.26	217.86	10496.2	2.03	2.29
1089.00	1094.10	0.80	11.34	5.80	658.9	2.22	5.49
1094.10	1099.20	3.19	26.88	6.30	2627.2	5.27	9.24
1099.20	1104.30	7.64	43.45	5.63	6303.6	8.52	13.72
1104.30	1109.40	8.87	45.69	5.10	7316.4	8.96	15.15
1109.40	1114.50	8.72	45.24	5.11	7195.0	8.87	15.04

Errors, Warnings and Notes

Warning - Divided flow computed for this cross-section.
Warning - The energy loss was greater than 1.0 ft (0.3 m), between the current and previous cross section. This may indicate the need for additional cross

Figure 8.12 Example of the Flow Distribution Type of Table

At the bottom of each of the cross section tables are two text boxes for displaying messages. The bottom text box is used to display the definition of the variables listed in the table. When the user presses the left mouse button over any data field, the description for that field is displayed in the bottom text box. The other text box is used to display any **Errors, Warnings, and Notes** that may have occurred during the computations for the displayed cross section.

Cross Section Table Options

Under the **Options** menu of the cross section table window, the user has the following options:

Include Interpolated XS's. This option allows the user to either view interpolated cross section output or not. Turning the "include interpolated XS's" option on (which is the default), allows interpolated sections to be selected from the river station box. Turning this option off gets rid of all the interpolated sections from the river station selection box, and only the user entered cross sections are displayed.

Units System For Viewing. This option allows the user to view the output in either English or Metric units. It does not matter whether the input data is in english or metric, the output can be viewed in either system.

Profile Tables

Profile tables are used to show a limited number of hydraulic variables for several cross sections. To display a profile table on the screen, select **Profile Table** from the **View** menu of the main HEC-RAS window. An example profile table is shown in Figure 8.13.

Profile Output Table - Standard Table 1

File Options Std. Tables User Tables Help

HEC-RAS Plan: pressure Reach: Easy Creek

River Sta.	Q Total	Min Ch El	W.S. Elev	Crit W.S.	E.G. Elev	E.G. Slope	Vel Chnl
	(cfs)	(ft)	(ft)	(ft)	(ft)	(ft/ft)	(ft/s)
6	8000.00	510.00	525.24		525.50	0.000481	4.23
6	10000.00	510.00	530.38		530.53	0.000190	3.38
6	25000.00	510.00	533.28		533.88	0.000629	6.82
5	8000.00	510.00	525.09	517.30	525.41	0.000553	4.50
5	10000.00	510.00	530.23	518.16	530.47	0.000260	3.93
5	25000.00	510.00	533.17	522.78	533.78	0.000642	6.86
3.5	Bridge						
2	8000.00	510.00	525.03	517.30	525.35	0.000563	4.53
2	10000.00	510.00	529.98	518.15	530.23	0.000273	3.98

Top width of the wetted cross section.

Figure 8.13 Example Profile Table

There are several standard table (Std. Tables) types available to the user. Some of the tables are designed to provide specific information at hydraulic structures (e.g., bridges and culverts), while others provide generic information at all cross sections. The standard table types available to the user are:

Standard Table 1. The is the default profile type of table. This table gives you a summary of some of the key output variables.

Standard Table 2. This is the second of the standary summary tables. This table provides information on the distribution of flow between the left overbank, main channel, and right overbank. This table also shows the friction losses, as well as contraction and expansion losses that occured between each section. Energy losses displayed at a particular cross section are for the losses that occurred between that section and the next section downstream.

Four XS Culvert. This standard table provides summary results for the four cross sections around each of the culverts in the model. The four cross sections are the two immediately downstream and upstream of the culvert. This table will list all of the culverts in the model for the selected reaches.

Culvert Only. This standard table provides hydraulic information about the culvert, as well as the inlet control and outlet control computations that were performed.

Six XS Bridge. This table provides summary results for the six cross sections that make up a normal bridge transition. The six cross sections include the two cross sections just downstream of the bridge; the two cross sections inside of the bridge; and the two cross sections just upstream of the bridge. The program will display results for all the bridges in the model within the selected reaches. When viewing this table, on accassion there will be no displayed results for the cross sections inside of the bridge. This occurrs only when the user has selected a bridge modeling approach that does not compute results inside of the bridge. This includes: Yarnells method; the contracted opening method; both pressure flow equations; and pressure and weir flow solutions.

Bridge Only. The bridge only table shows summary information specifically for bridges.

Bridge Comparison. The bridge comparison table shows the results for all of the user selected bridge modeling approaches that were computed during the computations. For example, the program can calculate low flow bridge hydraulics by four different methods. The resuling upstream energy for the user selected methods will be displayed in this table.

Multiple Opening. This table shows a limited number of output variables for each opening of a multiple opening river crossing.

Encroachment 1, 2, and 3. These three standard tables provide various types of output for the computations of floodway encroachments.

To view one of the types of tables, select the desired table type from the **Std. Tables** menu on the profile table. In addition to the various types of profile tables, the user can specify which plans, profiles and reaches to include in the table. The plans, profiles and reaches options are available from the **Options** menu on the profile plot.

The user also has the ability to turn the viewing of interpolated cross sections on or off. The default is to view all cross sections, including the interpolated ones. To prevent the interpolated sections from showing up in the table, de-select **Include Interpolated XS's** from the **Options** menu.

Another feature available to users is the ability to set the number of decimal places that will be displayed for any variable of the pre-defined tables. Once a pre-defined table is selected from the **Tables** menu, select **Standard Table # Dec Places** from the **Options** menu. A window will appear displaying the current number of decimal places for each variable. The user can change the number of decimal places to what ever they wish.

User's also have the ability to view profile output tables in either english or metric units. This is available from the **Options** menu on the profile tables. It does not matter whether the input data is in english or metric, the output can be viewed in either system.

User Defined Output Tables

A special feature of the profile tables is the ability for users to define their own output tables. User defined output tables are available by selecting **Define Table** from the **Options** menu of the profile table. When this option is selected, a window will appear, as shown in Figure 8.14. At the top of the window is a table for the user selected variable headings (Table Column Headings), the units, and the number of decimal places to be displayed for each variable. Below this table is a table containing all of the available variables that can be included in your user defined table. The variables are listed in alphabetical order. Below the list of variables is a message box that is used to display the definition of the selected variable.

To add variables to the column headings, simply double click the left mouse button while the mouse pointer is over the desired variable. The variable will be placed in the active field of the table column headings. To select a specific column to place a variable in, click the left mouse button once while the mouse pointer is over the desired table column field. To delete a variable from the table headings, double click the left mouse button while the mouse pointer is over the variable that you want to delete. The number of decimal places for each variable can be changed by simply typing in a new value.

User defined tables are limited to 15 variables. Once you have selected all of the variables that you want, press the **OK** button at the bottom of the window. The profile table will automatically be updated to display the new table.

Once you have the table displayed in the profile table window, you can save the table headings for future use. To save a table heading, select **Save Table** from the **Options** menu on the profile table window. When this option is selected, a popup window will appear, prompting you to enter a name for the table. Once you enter the name, press the **OK** button at the bottom of the popup window. The table name will then be added to a list of tables included under the **User Tables** menu on the profile table window. To delete a table from the list of user defined tables, select **Remove Table** from the **Options** menu of the profile table window. When this option is selected, a popup window will appear displaying a list of all the user defined tables. Click the

left mouse button over the tables that you want to delete, then press the **OK** button. The selected tables will then be deleted from the **User Tables** menu list.

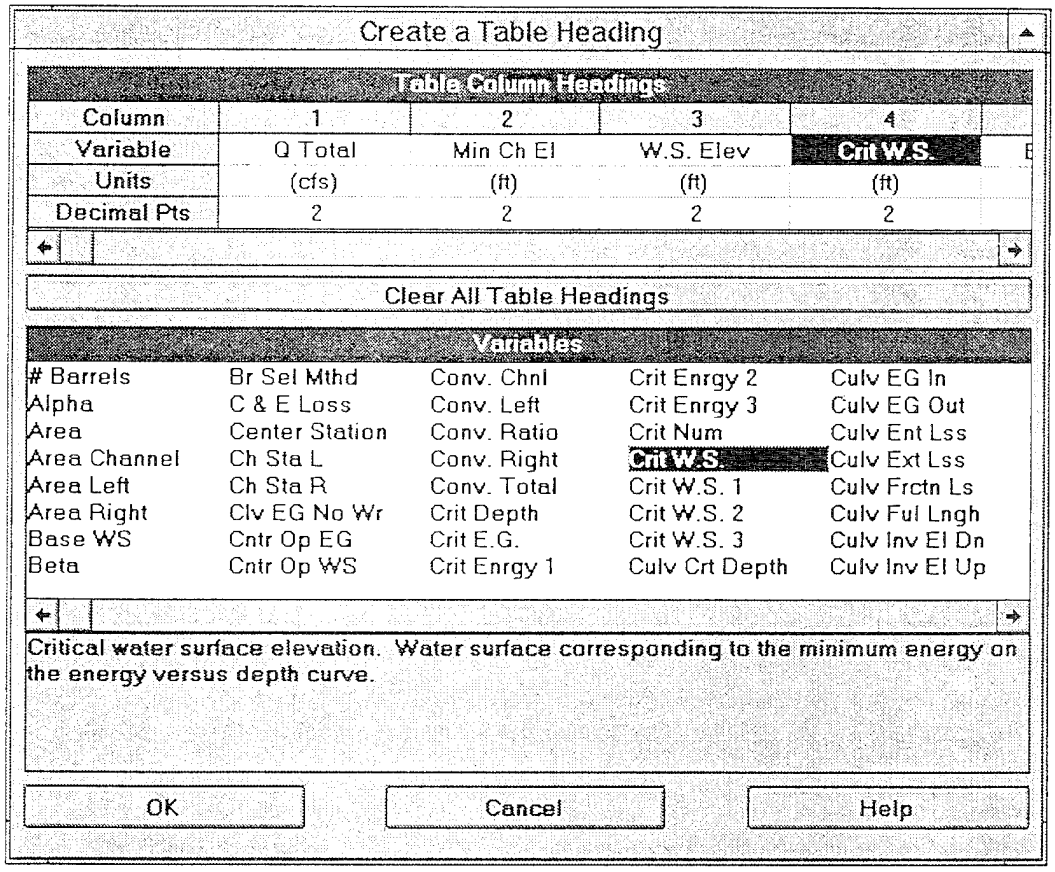


Figure 8.14 User Defined Tables Window

Sending Tables to the Printer

To send a table to the printer do the following:

1. Bring up the desired table from the tabular output (cross section or profile tables) section of the program.

2. Select **Print** from the **File** menu of the displayed table. When this option is selected, a popup window will appear allowing you to modify the default print options. Once you have set the printer with the desired options, press the **Print** button. The table will be sent to the Windows Print Manager. The Windows Print Manager will control the printing of the table.

The profile type of tables allow you to print a specific portion of the table, rather than the entire table. If you desire to only print a portion of the table, do the following:

1. Display the desired profile type table on the screen.

2. Using the mouse, press down on the left mouse button and highlight the area of the table that you would like to print. To get an entire row or column, press down on the left mouse button while moving the pointer across the desired row or column headings.

3. Select **Printer** from the **File** menu of the displayed table. Only the highlighted portion of the table and the row and column headings will be sent to the Windows Print Manager.

Sending Tables to the Windows Clipboard

To pass a table to the Windows Clipboard, and then to another program, do the following:

1. Display the desired table on the screen.

2. Select **Copy to Clipboard** from the **File** menu of the displayed table.

3. Bring up the program that you want to pass the table into. Select **Paste** from the **Edit** menu of the receiving program.

Portions of the profile tables can be sent to the Clipboard in the same manner as sending them to the printer.

Viewing Results From the River System Schematic

The user has the option of either bringing up graphics and tables from the **View** menu on the main HEC-RAS window (as discussed above), or from the river system schematic (found under geometric data). Once data have been entered, and a successful simulation has been made, the user can interact with the river system schematic. When the left mouse button is pressed over the river system schematic, a popup menu will appear listing options that are relevant to the area of the schematic that is located under the mouse pointer. An example of this is shown in Figure 8.15.

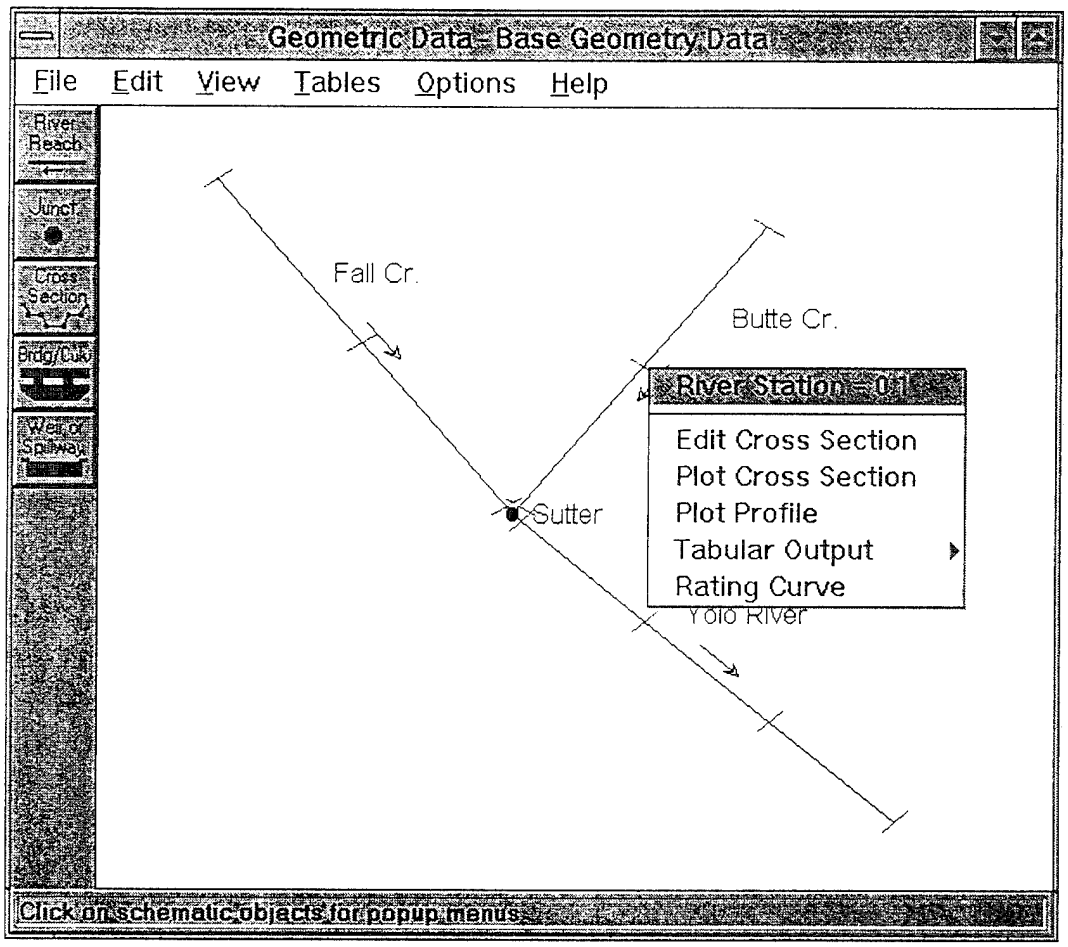


Figure 8.15 Geometric Data Window With Popup Menu

In Figure 8.15, the popup menu shown comes up whenever the user presses the left mouse button over a cross section. In this particular example, the mouse button was pressed over the cross section located at river station 0.1 of the Butte Cr. river reach. As shown in the menu, the user has the choice of

editing the cross section data; plotting the cross section; plotting the profile for the reach containing this cross section; viewing tabular output; or plotting the computed rating curve at this cross section. Other popup menus are available for bridges; culverts; junctions; and reach data.

CHAPTER 9

Performing a Floodway Encroachment Analysis

The evaluation of the impact of floodplain encroachments on water surface profiles can be of substantial interest to planners, land developers, and engineers. It is also a significant aspect of flood insurance studies. HEC-RAS contains five optional methods for specifying floodplain encroachments. This chapter describes how to enter floodway encroachment data, how to perform the encroachment calculations, and viewing the floodway encroachment results.

For information on the computational details of each of the five encroachment methods, as well as special considerations for encroachments at bridges, culverts, and multiple openings, see Chapter 8 of the HEC-RAS hydraulics reference manual.

Contents

- General
- Entering Floodway Encroachment Data
- Performing the Floodway Encroachment Analysis
- Viewing the Floodway Encroachment Results

General

The HEC-RAS floodway procedure is based on calculating a natural profile (existing conditions geometry) as the first profile in a multiple profile run. Other profiles, in a run, are calculated using various encroachment options, as desired. Before performing an encroachment analysis, the user should have developed a model of the existing river system. This model should be calibrated to the fullest extent that is possible. Verification that the model is adequately modeling the river system is an extremely important step before attempting to perform an encroachment analysis.

Entering Floodway Encroachment Data

Within HEC-RAS, the data for performing a floodway encroachment analysis are entered from the Steady Flow Analysis window. Encroachment information is not considered as permanent geometry or flow data, and is therefore not entered as such. The enchoachment information is saved as part of the existing Plan data.

To bring up the floodway encroachment data window, select the **Encroachments** option from the **Options** menu of the Steady Flow Analysis window. When this option is selected an Encroachment window will appear as shown in Figure 9.1 (except yours will be blank when you first bring it up).

As shown in Figure 9.1, There are several pieces of data that the user must supply for an encroachment analysis. The encroachment analysis can only be performed for profiles 2 through 15 (or what ever number has been set by the user in the flow data editor. Encroachments are not performed on profile one because most of the encroachment methods rely on having a base profile for comparison.

The data for an encroachment analysis should be entered in the following manner:

Global Information. Global information are data that will be applied at every cross section for every profile computed. The first piece of global information is the **Equal Conveyance Reduction** selection box at the top of the Encroachment data editor window. Equal conveyance reduction applies to encroachment methods 3, 4, and 5. When this is turned on, the program will attempt to encroach, such that an equal loss of conveyance is provided on both sides of the stream. If this option is turned off, the program will encroach by trying to maintain a loss in conveyance in proportion to the distribution of natural overbank conveyance. The default is to have equal conveyance reduction turned on.

The second item under global information, is the **Left bank offset** and the **Right bank offset**. The left and right offset is used to establish a buffer zone around the main channel for further limiting the amount of the encroachments. For example, if a user established a right offset of 5 feet and a left offset of 10 feet, the model will limit all encroachments to 5 feet from the left bank station and 10 feet from the right bank station. The default is to have no left or right offset, this will allow the encroachments to go up to the main channel bank stations, if necessary.

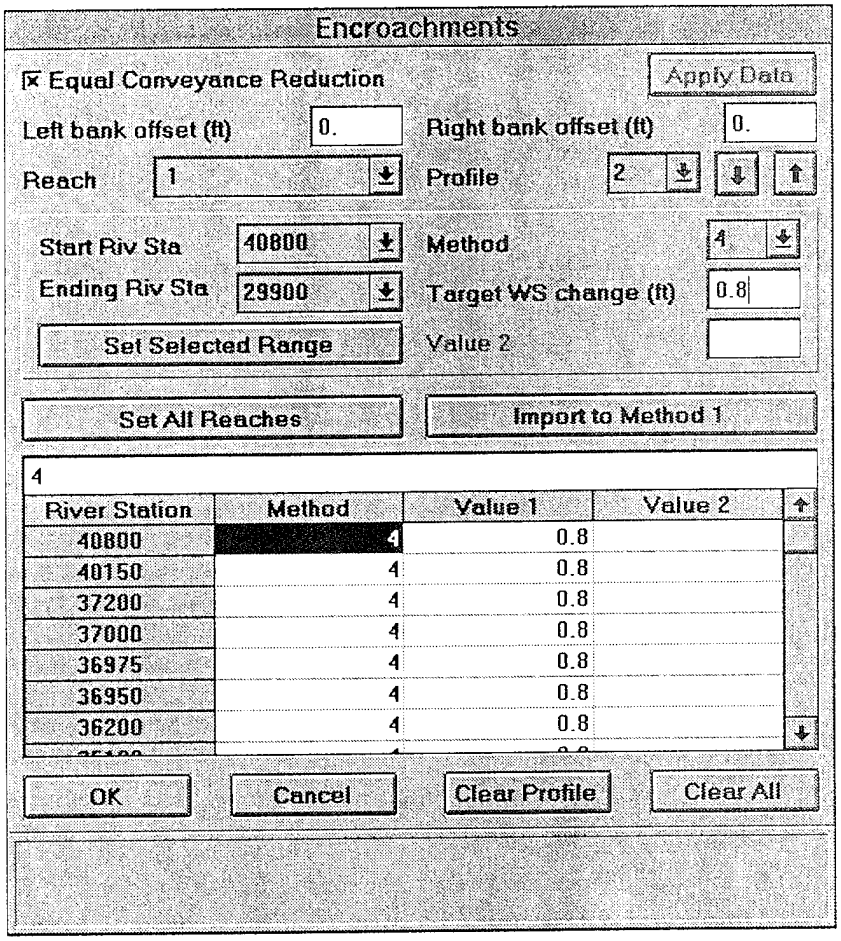


Figure 9.1 Floodway Encroachment Data Editor

Reach and River Station Selection Boxes. The next piece of data for the user to select, is the reach in which to enter encroachment data. The user is limited to seeing one reach at a time on the encroachment data editor. Once a reach is selected, the user can then enter a **Starting and Ending River Station** to work on. By default, the program selects all the sections in the reach. The user can change this to any range of cross sections within the reach.

Profile. Next, the user should select a profile number to work on. Profiles

are limited to 2 though the maximum number set in the currently opened flow data (e.g. 2 through 4, if the user has set 4 profiles in the flow data editor). The user can not set encroachments for profile 1.

Method and Target Values. The next step is to enter the desired encroachment method to be used for the currently selected profile. Once a method is selected, the data entry boxes that corresponds to that method will show up below the method selection box. Some of the methods require only one piece of data, while others require two. The user should then enter the required information that corresponds to the method that they have selected. For example, if the user selects encroachment method 4, only one piece of information is required, the target change in water surface elevation. The available encroachment methods in HEC-RAS are:

Method 1 -	User enters right and left encroachment station
Method 2 -	User enters a fixed topwidth
Method 3 -	User specifies the percent reduction in conveyance
Method 4 -	User specifies a target water surface
Method 5 -	User specifies target water surface and maximum change in energy

Set Selected Range. Once the encroachment method is selected, and its corresponding data are entered, the user should press the **Set Selected Range** button. Pressing this button will fill in the table below with the selected range of river stations; the selected method; and the corresponding data for the method. Note that, if the selected method only has one data item, that method's data will go under the **Value 1** column of the table. If the selected method has two data items, the first goes into the **Value 1** column and the second goes into the **Value 2** column. Once the data is put into the table, the user can change the method and corresponding data values directly from the table.

At this point the user should repeat these tasks until all of the encroachment data are entered (i.e. for all the reaches and locations in the model, as well as all of the profiles for which the user wants to perform the encroachment analysis). Once all of the encroachment data are entered, the user presses the **OK** button and the data will be applied and the window will close. The user can return to the encroachment window and edit the data at any time. The encroachment data are not saved to the hard disk at this time, they are only saved in memory. To save the data to the hard disk, the user should either select **Save Project** from the File menu of the main HEC-RAS window, or select **Save Plan** from the File menu of the Steady Flow Analysis window.

Some additional options on the Encroachment data entry window include: **Set All Reaches** and **Import Method 1**. The **Set All Reaches** option allows the user to set the selected method and corresponding data for all reaches and locations in the model (all cross sections, bridges, culverts, etc..). This is a

quick way to start a preliminary encroachment analysis without having to enter a lot of data. The **Import Method 1** option, allows the user to transfer the computed encroachment stations from a previous run to the input data for a future run. For example, if the user performs a preliminary encroachment analysis using any of the methods 2 through 5, they may want to convert the results from one of the runs to a method 1 encroachment method. This will allow the user to further define the floodway, using method 1, without having to enter all of the encroachment stations. The import of encroachment stations, in this manner, is limited to the results of a single encroachment profile for each reach.

Performing The Floodway Encroachment Analysis

The HEC-RAS floodway procedure is based on calculating a natural profile (no encroachments) as the first profile of a multiple profile run. Subsequent profiles are calculated with the various encroachment options available in the program.

In general, when performing a floodway analysis, encroachment methods 4 and 5 are normally used to get a first cut at the encroachment stations. Recognizing that the initial floodway computations may provide changes in water surface elevations greater, or less, than the "target" increase, initial computer runs are usually made with several "target" values. The initial computer results should then be analyzed for increases in water surface elevations, changes in velocities, changes in top width, and other parameters. Also, plotting the results with the X-Y-Z perspective plot, or onto a topo map, is recommended. From these initial results, new estimates can be made and tested.

After a few initial runs, the encroachment stations should become more defined. Because portions of several computed profiles may be used, the final computer runs are usually made with encroachment Method 1 defining the specific encroachment stations at each cross section. Additional runs are often made with Method 1, allowing the user to adjust encroachment stations at specific cross sections to further define the floodway.

While the floodway analysis generally focuses on the change in water surface elevation, it is important to remember that the floodway must be consistent with local development plans and provide reasonable hydraulic transitions through the study reach. Sometimes the computed floodway solution, that provides computed water surfaces at or near the target maximum, may be unreasonable when transferred to the map of the actual study reach. If this occurs, the user may need to change some of the encroachment stations, based on the visual inspection of the topo map. The floodway computations should be re-run with the new encroachment stations to ensure that the target maximum is not exceeded.

Viewing the Floodway Encroachment Results

Floodway encroachment results can be viewed in both graphical and tabular modes. Graphically, the encroachment results show up on the cross section plots as well as the X-Y-Z Perspective plot. An example cross section plot is shown in Figure 9.2.

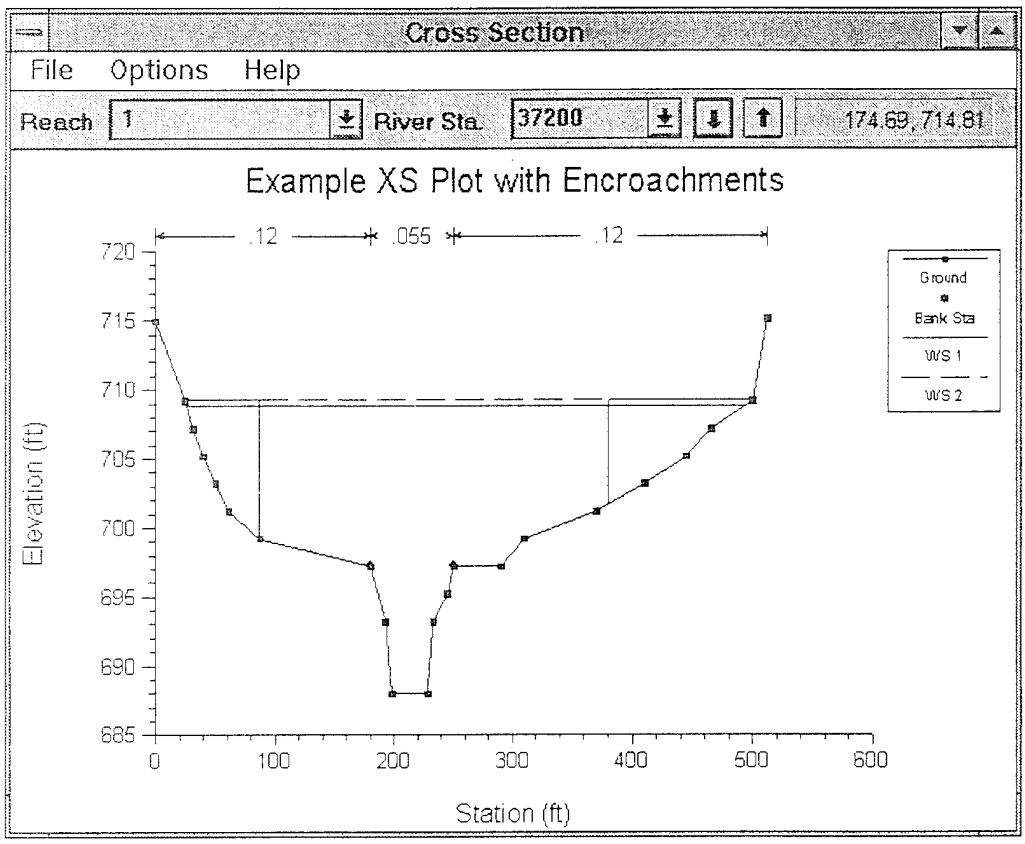


Figure 9.2 Example Cross Section Plot With Encroachments

As shown in Figure 9.2, the encroachments are plotted as outlined blocks. In this example, the water surface profile for the base run (first profile) is plotted along with one of the encroached profiles. The user can plot as many profiles as they wish, but it may become a little confusing with several sets of encroachments plotted at they same time.

Another type of graphic that can be used to view the encroachments is the X-Y-Z perspective plot, an example is shown in Figure 9.3. In this example, the base profile (profile 1) as well as one of the encroached profiles is plotted at the same time over a range of cross sections. This type of plot allows the user to get a reach view of the floodway encroachment. The user can quickly see if the encroachments transition smoothly or if they are erratic. In general, the final encroachments should have a consistent and smooth transition from one

cross section to the next. With the assistance of this type of plot, the user may want to further refine the final encroachment stations and re-run the model.

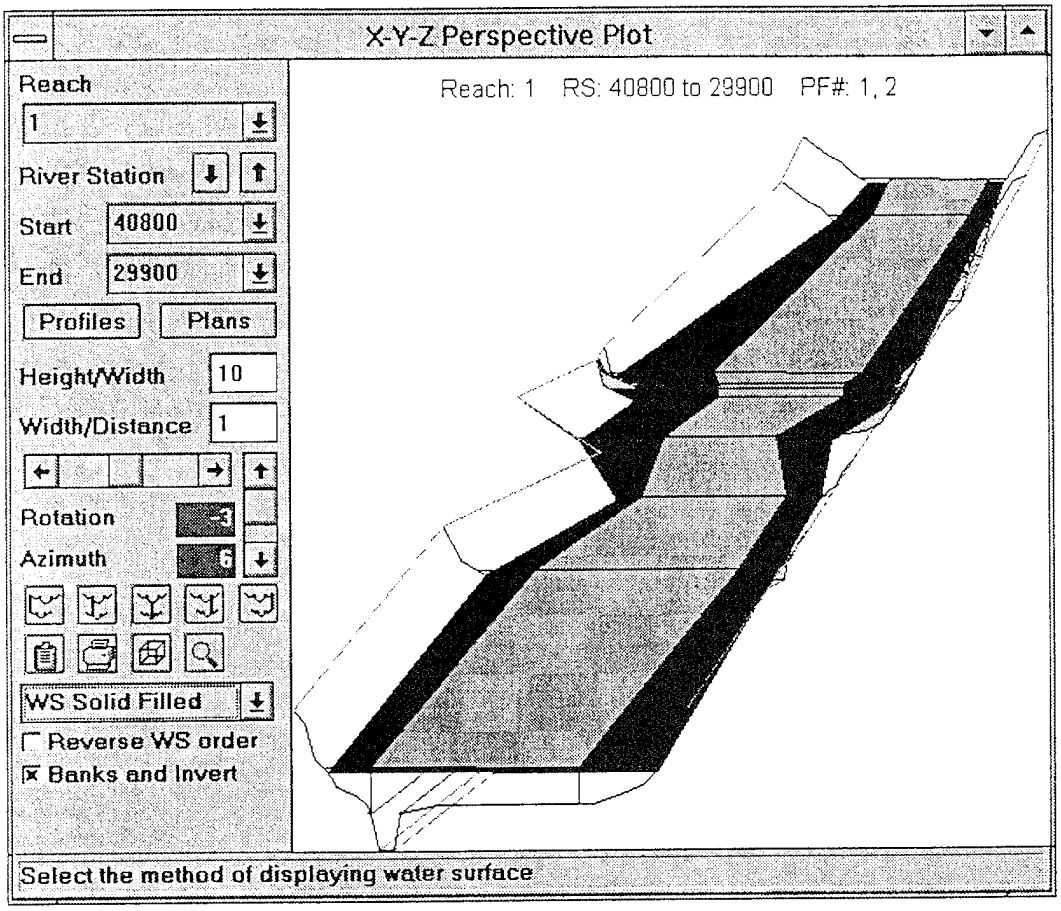


Figure 9.3 Example X-Y-Z Perspective Plot with Base and Encroached Profiles

Encroachment results can also be viewed in a tabular mode from the Profile Output Tables. Select **Profile Table** from the **View** menu of the main HEC-RAS window. When the table comes up, the user can select from three different pre-defined encroachment tables. To bring up one of the encroachment tables, select **Encroachment 1** from the **Std. Tables** menu on the Profile table window. An example of Encroachment 1 table is shown in Figure 9.4. The table shows the basic encroachment results of : computed water surface elevation; change in water surface from the base profile; the computed energy; top width of the active flow area; the flow in the left overbank, main channel, and right overbank; the left encroachment station; the station of the left bank of the main channel; the station of the right bank of the main channel; and the right encroachment station.

Profile Output Table - Encroachment 1

File Options Std. Tables User Tables Help

HEC-RAS Plan: Floodway Reach: 1

River Sta.	W.S. Elev	Prof Delta	E.G. Elev	Top Wdth Act	Q Left	Q Channel	Q Right	Enc Sta L
	(ft)	(ft)	(ft)	(ft)	(cfs)	(cfs)	(cfs)	(ft)
36200	705.03		705.33	400.52	1446.64	3987.24	2566.13	
36200	705.91	0.88	706.26	187.97	1174.11	4307.82	2518.07	182.98
36200	706.10	1.08	706.46	178.81	1103.29	4392.29	2504.42	187.47
36200	706.48	1.45	706.84	170.70	1032.90	4466.95	2500.15	191.45
35100	703.73		704.04	324.35	528.34	3498.08	3973.58	
35100	704.66	0.92	704.99	241.01	109.85	3776.00	4114.15	108.95
35100	704.84	1.11	705.19	230.18	3.32	3854.03	4142.65	114.37
35100	705.20	1.46	705.47	219.63		3530.57	4469.43	115.00
33700	702.17		702.40	465.48	2091.96	3489.63	2418.42	
33700	703.16	0.99	703.41	303.58	1921.26	3741.27	2337.47	127.48
33700	703.33	1.16	703.59	290.54	1868.19	3824.11	2307.70	131.67
33700	703.50	1.32	703.77	274.79	1816.28	3906.37	2277.35	138.67
29900	698.30		698.50	526.75	4001.82	3654.32	343.86	
29900	699.30	1.00	699.49	353.21	4287.34	3712.66		164.79
29900	699.30	1.00	699.51	334.71	4167.98	3832.02		173.29
29900	699.30	1.00	699.52	317.26	4047.86	3952.14		190.74

Right station of encroachment.

Figure 9.4 Example of the Encroachment 1 Standard Table

Encroachment 2 table provides some additional information that is often used when plotting the encroachments onto a map. This table includes: the change in water surface elevations from the first profile; the top width of the active flow area; the percentage of conveyance reduction in the left overbank; the left encroachment station; the distance from the center of the main channel to the left encroachment station; the station of the center of the main channel; the distance from the center of the main channel to the right encroachment station; the right encroachment station; and the percentage of conveyance reduction in the right overbank. An example of the Encroachment 2 standard table is shown in Figure 9.5.

Profile Output Table - Encroachment 2

File Options Std. Tables User Tables Help

HEC-RAS Plan: Floodway Reach: 1

River Sta.	Prof Delta	Top Wdth Act	K Perc L	Enc Sta L	Dist Center L	Center	Dist Center R	Enc Sta R
	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)	(ft)
40800		216.60				120.00		
40800	1.00	125.00	2.91	95.00	25.00	120.00	100.00	220.00
40800	1.18	117.15	3.00	95.00	25.00	120.00	92.15	212.15
40800	1.35	109.21	3.10	95.00	25.00	120.00	84.21	204.21
40150		245.98				120.00		
40150	0.75	141.93	3.79	95.00	25.00	120.00	116.93	236.93
40150	0.92	132.27	3.88	95.00	25.00	120.00	107.27	227.27
40150	1.12	125.00	3.98	95.00	25.00	120.00	100.00	220.00
37200		467.07				215.00		
37200	0.45	293.23	4.87	86.90	128.10	215.00	165.13	380.13
37200	0.59	282.11	6.21	87.90	127.10	215.00	155.00	370.00
37200	0.80	271.04	7.30	94.31	120.69	215.00	150.35	365.35
37000		465.11				215.00		
37000	0.38	240.64	6.01	100.12	114.88	215.00	125.76	340.76
37000	0.52	222.42	7.35	108.38	106.62	215.00	115.80	330.80
37000	0.72	206.10	8.63	115.84	99.16	215.00	106.93	321.93

Figure 9.5 Example of the Encroachment 2 Standard Table

The last encroachment table, Encroachment 3, provides the minimum floodway data for reporting. This table includes: the active flow top width; the flow area (including any ineffective flow area); the average velocity of the entire cross section; the computed water surface elevation; the base water surface elevation (profile 1); and the change in water surface from the first profile. An example of this table is shown in Figure 9.5

Profile Output Table - Encroachment 3

File Options Std. Tables User Tables Help

HEC-RAS Plan: Floodway Reach: 1

River Sta.	Top Wdth Act	Area	Vel Total	W.S. Elev	Base WS	Prof Delta
	(ft)	(sq ft)	(ft/s)	(ft)	(ft)	(ft)
40800	216.60	1698.56	4.71	711.93	711.93	
40800	125.00	1443.99	5.54	712.93	711.93	1.00
40800	117.15	1396.15	5.73	713.10	711.93	1.18
40800	109.21	1344.40	5.95	713.28	711.93	1.35
40150	245.98	2136.39	3.74	710.82	710.82	
40150	141.93	1811.67	4.42	711.58	710.82	0.75
40150	132.27	1744.74	4.59	711.74	710.82	0.92
40150	125.00	1695.75	4.72	711.94	710.82	1.12
37200	467.07	4273.76	1.87	708.81	708.81	
37200	293.23	3608.96	2.22	709.26	708.81	0.45
37200	282.11	3561.58	2.25	709.41	708.81	0.59
37200	271.04	3513.40	2.28	709.62	708.81	0.80
37000	465.11	4229.24	1.89	708.72	708.72	
37000	240.64	3106.03	2.58	709.10	708.72	0.38
37000	222.42	2961.37	2.70	709.23	708.72	0.52
37000	206.10	2839.59	2.82	709.43	708.72	0.72
36950	115.62	3008.65	2.66	705.93	705.93	

Figure 9.6 Example of the Encroachment 3 Standard Table

CHAPTER 10

Trouble Shooting With HEC-RAS

The HEC-RAS software is designed to continue its computations all the way through completion, even when the user has entered poor data. Because of this, the fact that the program executes a complete run does not necessarily mean that the results are good. The user must carefully review the results to ensure that they adequately represent the study reach and that they are reasonable and consistent. The HEC-RAS software is an engineering tool, it is by no means a replacement for sound engineering.

The HEC-RAS software contains several features to assist the user in the development of a model; debugging problems; and the review of results. These features include: built in data checking; an Errors, Warnings, and Notes system; and a computational Log Output file. In addition to these features, the user can use the graphical and tabular output to review the results and check the data for reasonableness and consistency.

Built in Data Checking

The HEC-RAS user interface has two types of built in data checking. The first type of data checking is performed as the user enters the data. Each data field of the data entry editors has some form of data checking. The second type of data checking occurs when the user starts the steady flow computations. When the user presses the compute button, on the steady flow analysis window, the entire data set is processed through several data checks before the computations are processed. A detailed discussion of each of these two data checking features is described below.

Checking the Data as it is Entered

This type of data checking occurs whenever the user enters data into a single data field or table. Once the user leaves a particular data entry field or table, the program will automatically check that data for reasonableness. The following is a list of some of the types of data checks that are performed:

1. Minimum and maximum range checking for variables.

2. Alpha and numeric data checks. This is done to ensure that the right type of data is entered in each field.

3. Increasing order of station for cross sections, bridge deck/roadway, and abutments.

4. Data consistency checks (i.e. when the main channel bank stations are entered, the program checks to see if they exist in the cross section station and elevation data).

5. Data deletion warnings. When you delete data the software will give you a warning before it is deleted.

6. File management warnings (i.e. program will give you a chance to save the data to the hard disk before the program is closed, or a different data set is opened).

7. Data geometry checks (i.e. when a bridge deck/roadway is entered, the program checks to ensure that the deck/roadway intersects with the ground data).

Data Checking Before Computations are Performed

The second type of data checking is performed to evaluate the completeness and consistency of the data. This type of data checking occurs before the computations take place. When the user presses the **Compute** button on the Steady Flow Analysis window, the program will perform a series of data checks before the computations are allowed to proceed. If any data errors are found, the program will not perform the computations. The following is a list of some of the types of checks that are made during this time:

1. Data completeness. These data checks insure that all of the required data exists for the entire data set. If any missing data are found, a complete list of all the missing data and their specific locations is displayed on the screen. An example of this is shown in Figure 10.1.

2. Data consistency. This type of data checking is performed to ensure that the data is consistent with the computations that are being requested. For example, if the user asks to perform a mixed flow regime computation, the program checks to ensure that upstream as well as downstream boundary conditions have been specified. Likewise, if an encroachment analysis is requested, the program checks to ensure that the number of profiles lines up with the number specified in the encroachment data. There are several other checks of this type.

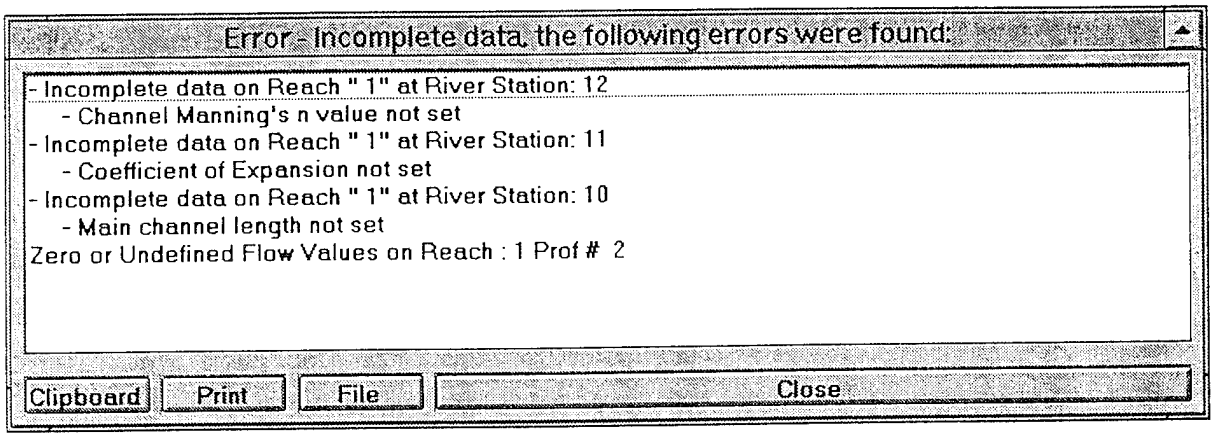


Figure 10.1 Data Completeness Checking Window

Errors, Warnings, and Notes

The HEC-RAS software has a system of Errors, Warnings, and Notes that are passed from the steady flow computation program to the user interface. During the computations, the steady flow computation program will set flags for at a particular node (nodes are cross sections, bridges, culverts, or multiple openings) whenever it is necessary. These message flags are written to the standard output file, along with the computed results for that node. When the user interface reads the computed results from the output file, if any errors, warnings, or notes exist, they are interpreted and displayed in various locations from the interface.

The user can request a summary of all the errors, warnings, and notes that occurred during the computations. This is accomplished by selecting **Summary Errors, Warnings, and Notes** from the **View** menu on the main HEC-RAS window. Once this is selected, a window will pop up displaying all of the messages. The user has the options of expanding the window; printing the messages; or sending them to the windows clipboard. An example of the Errors, Warnings, and Notes window is shown in Figure 10.2.

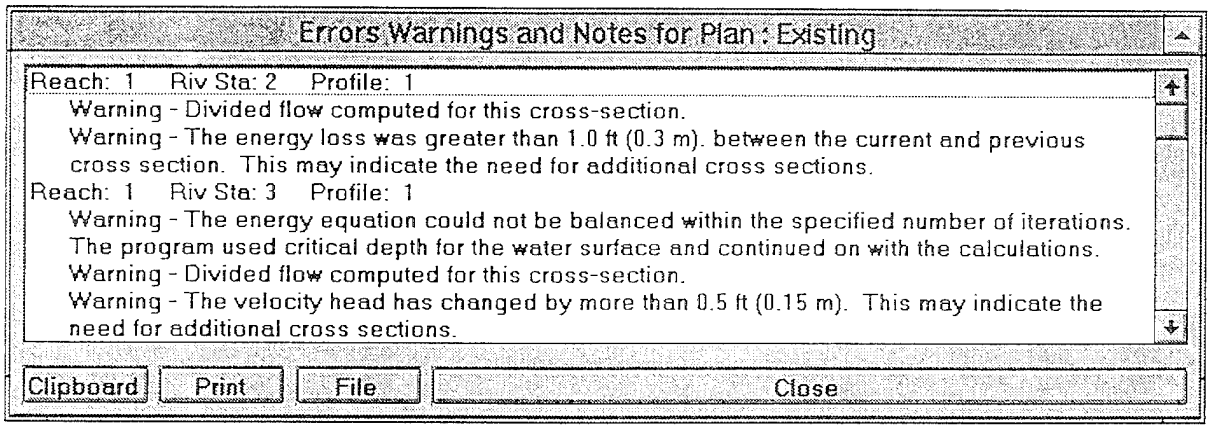


Figure 10.2 Summary of Errors, Warnings, and Notes Window

Besides the summary window, errors, warnings, and notes will automatically appear on the cross section specific tables. When a specific cross section or hydraulic structure is being displayed, any errors, warnings, or notes that were set at that location, for the displayed profile, will show up in the Errors, Warnings, and Notes message box at the bottom of the table. An example of this table is shown in Figure 10.3.

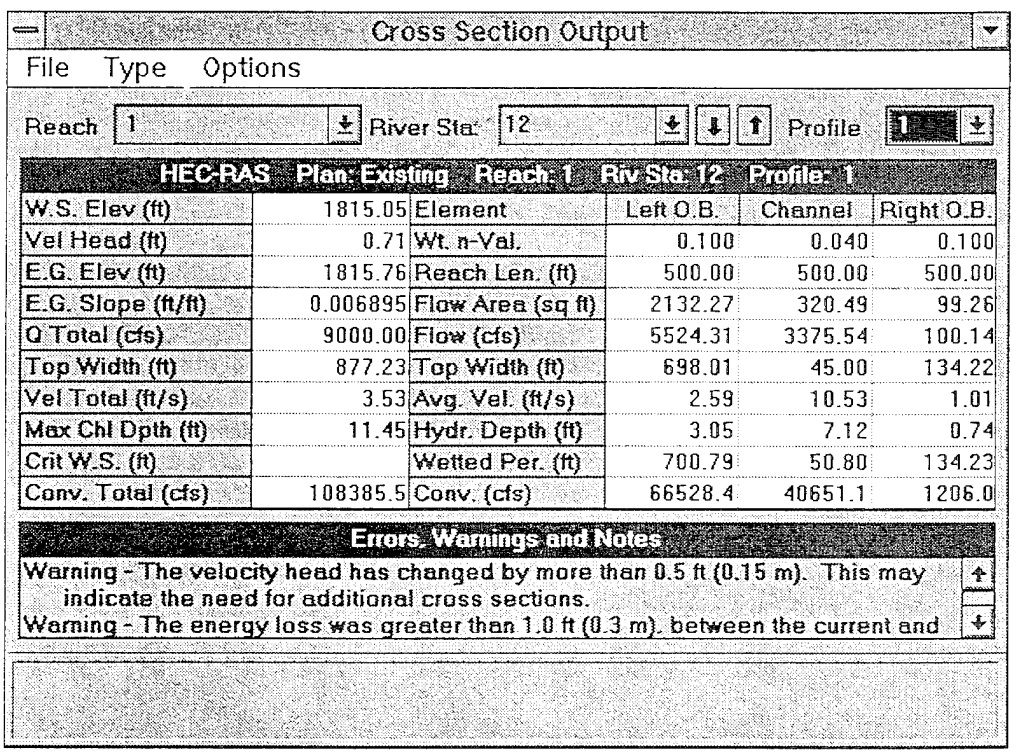


Figure 10.3 Cross Section Table with Errors, Warnings, and Notes

In general, the errors, warnings, and notes messages should be self explanatory. The three categories of messages were developed to represent the following:

ERRORS: Error messages are only sent when there are problems that prevent the program from being able to complete the run.

WARNINGS: Warning messages provide information to the user that may or may not require action on the user's part. In general, whenever a warning is set at a location, the user should review the hydraulic results at that location to ensure that the results are reasonable. If the hydraulic results are found to be reasonable, then the message can be ignored. However, in many instances, a warning level message may require the user to take some action that will cause the message to disappear on future runs. Many of the warning messages are caused by either inadequate or bad data. Some common problems that cause warning messages to occur are the following:

Cross sections spaced to far apart. This can cause several warning messages to be set.

Cross sections starting and ending stations not high enough. If a computed water surface is higher than either end point of the cross section, a warning message will appear.

Bad Starting Water Surface Elevation. If the user specifies a boundary condition that is not possible for the specified flow regime, the program will take action an set an appropriate warning message.

Bad Cross Section Data. This can cause several problems, but most often the program will not be able to balance the energy equation and will default to critical depth.

NOTES: Note level messages are set to provide information to the user about how the program is performing the computations.

Log Output

Setting Log File Output Level

This option allows the user to set the level of the Log file. The Log file is a file that is created by the computational program. This file contains information tracing the program process. Log levels can range between 0 and 10, with 0 resulting in no Log output and 10 resulting in the maximum Log

output. In general, the Log file output level should not be set unless the user gets an error during the computations. If an error occurs in the computations, set the log file level to an appropriate value. Re-run the computations and then review the log output, try to determine why the program got an error.

When the user selects **Set Log File Output Level**, a window will appear as shown in Figure 10.4. The user can set a "Global Log Level", which will be used for all cross sections and every profile. The user can also set log levels at specific locations for specific profiles. In general, it is better to only set the log level at the locations where problems are occurring in the computations. To set the specific location log level, first select the desired reach and river station. Next select the log level and the profile number (the log level can be turned on for all profiles). Once you have everything set, press the **Set** button and the log level will show up in the window below. Log levels can be set at several locations individually. Once all of the Log Levels are set, press the **OK** button to close the window.

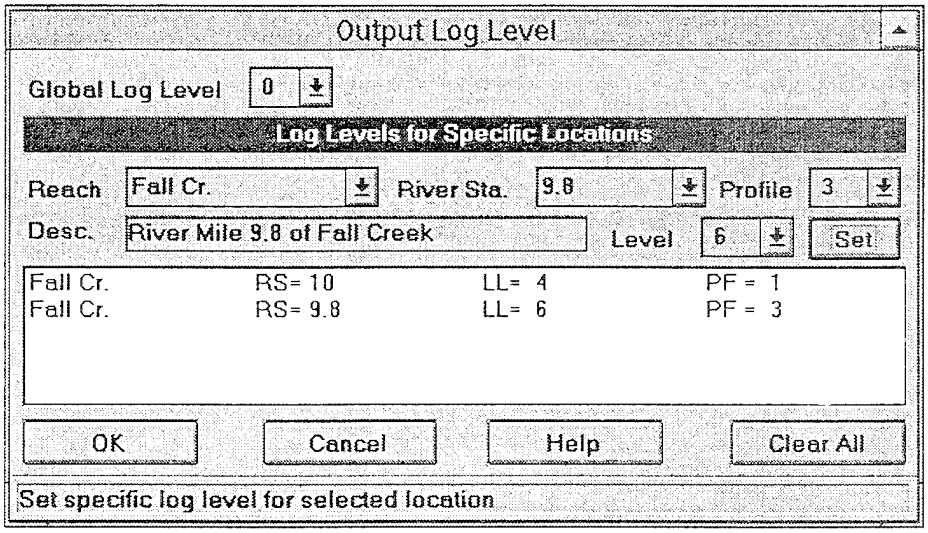


Figure 10.4 Log File Output Level window

Warning !!! - setting the global log output level to 4 or 5 can result in very large log file output. Global log level values of 6 or larger can result in extremely large log files.

Viewing The Log File

This option allows the user to view the contents of the log file. The interface uses the Windows Write program to accomplish this. It is up to the user to set an appropriate font in the Write program. If the user sets a font that uses proportional spacing, the information in the log file will not line up correctly. Some fonts that work well are: Line Printer; Courier (8 pt.); and Helvetica (8 pt.). Consult your Windows user's manual for information on how to use the Write program.

Reviewing and Debugging the Normal Output

After the user has successfully completed a run, and reviewed all the errors, warnings, and notes, the normal output should be reviewed for consistency and reasonableness.

Viewing Graphics

In general, the graphical output should be used as much as possible to get a quick view of the results. The user should look at all of the **cross sections** with the cross section plotting capability. The cross section plots will assist the user in finding data mistakes, as well as possible modeling mistakes (mistakes in ineffective flow areas, levees, n values, etc...).

The **profile plotting** capability is also a way to get a quick overview of the entire study area. The user should look for sudden changes to the energy grade line and the water surface. In general, these two variables should transition smoothly along the channel. If the user finds rapid changes in the energy or the water surface, the results at those locations should be reviewed closely to ensure that they are correct.

The **X-Y-Z Perspective Plot** can also be used to get a quick view of an entire reach. This plot is very helpful for viewing the top width of the flow area. If the user finds dramatic changes in the top width from one cross section to the next, then the results at those locations should be reviewed closely. Dramatic changes in top width may indicate the need for additional cross sections.

Viewing Tabular Output

There are several types of tabular output. The user should try to make use of all of them when viewing tabular results. In general, the profile types of tables should be used to get an overview of some of the key variables at several locations. If any problems are found, or any results that seem suspect, the user should use the cross section specific tables to get detailed results at a single location.

The Occurrence of Critical Depth

During the water surface profile calculations, the program may default to critical depth at a cross section in order to continue the calculations. Critical depth can occur for the following reasons:

1. Bad cross section data: If the energy equation can not balance because of bad cross section data, the program defaults to critical depth.

2. Cross sections spaced to far apart: If the cross sections are spaced to far apart, the program may not be able to calculate enough energy losses to obtain a subcritical water surface at the upstream section.

3. Wrong flow regime: When calculating a subcritical profile, and the program comes to a reach that is truly supercritical, the program will default to critical depth. Likewise, when calculating a supercritical profile, if the reach is truly subcritical, the program will default to critical depth.

4. Program can not balance the energy equation above or below the top of a levee or ineffective flow area: On occasion, when the program is balancing a water surface that is very close to the top of a levee, or an ineffective flow area, the program may go back and forth (above and below the levee) without being able to balance the energy equation. When this occurs, the program will default to critical depth.

Appendix A References

Barkau, Robert L., 1992. *UNET, One-Dimensional Unsteady Flow Through a Full Network of Open Channels*, Computer Program, St. Louis, MO.

Federal Highway Administration, 1978. *Hydraulics of Bridge Waterways*, Hydraulic Design Series No. 1, by Joseph N. Bradley, U.S. Department of Transportation, Second Edition, revised March 1978, Washington D.C..

Federal Highway Administration, 1985. *Hydraulic Design of Highway Culverts*, Hydraulic Design Series No. 5, U.S. Department of Transportation, September 1985, Washington D.C..

Hydrologic Engineering Center, 1991. *HEC-2, Water Surface Profiles*, User's Manual, U.S. Army Corps of Engineers, Davis CA.

Hydrologic Engineering Center, 1993. *UNET, One-Dimensional Unsteady Flow Through a Full Network of Open Channels*, User's Manual, U.S. Army Corps of Engineers, Davis, CA.

Hydrologic Engineering Center, 1994. HECDSS, User's Guide and Utility Programs Manual, U.S. Army Corps of Engineers, Davis CA.

Microsoft Corporation, 1992. *Microsoft Windows 3.1*, User's Manual, Redmond WA.

U.S. Army Corps of Engineers, 1965. *Hydraulic Design of Spillways*, EM 1110-2-1603, Plate 33.